

2025 ROUTER-CIM AUTOMATION SUITE

CNC Software Solutions by CIM-TECH

CONTACT US



877-549-8211



cim-tech.com



CIM-TECH
AUTOMATED CAD/CAM SOLUTIONS



by CIM-TECH.COM, Inc.

We are pleased to announce the release of Router-CIM Automation Suite. This state-of-the-art programming software combines the latest advances in CNC machine tool programming with the industry-standard CAD features found only in AutoCAD®, the world's premier Computer Aided Design software.

Router-CIM Automation Suite

© 2025 CIM-TECH.COM, Inc.

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

Published: July 2025

Applied Topics Include:

Router-CIM

Router-CIM Macros

Geoshape and Cut

Expert Nurbs Cutter

Pocketing

Profile cutting

Drilling

Configuration

Modification

Knowledge

*CIM-TECH.COM, Inc.
7512 Dr. Phillips Blvd.
Suite 50-859
Orlando, FL 32819
United States
www.cim-tech.com*

*Toll-Free Technical Support:
(877) 549-8211*

Table of Contents

Part I	System Requirements and Installation	15
1	Product Overview	16
	Installation Quick Start	17
2	Critical Issues.....	17
3	Installation Guide.....	18
	Installing On A Different Computer	31
4	Updates.....	32
5	Uninstall.....	32
Part II	Router-CIM	34
1	Router-CIM in AutoCAD.....	34
	Router-CIM Toolbar	35
2	Configuration Wizard.....	37
	Instructions	38
	Knowledge	39
	Database	40
	Create Knowledge Base.....	41
	Import Knowledge Base.....	41
	Delete Knowledge Base.....	42
	Save Knowledge Base.....	43
	Default Knowledge Base.....	44
	Measurements	44
	Show Wizard	45
	Help	46
	How to configure Router-CIM for your machine	47
	Customizing the Knowledge Base	48
3	Router-CIM Control Panel.....	50
4	Geoshape, Group, and Start Point Edit.....	52
	Geoshape	53
	Start Point Edit (Start Pt)	54
	Open Shape Issues	56
	RCIM Group Command	57
	Troubleshooting Geometry	64
5	Tool Information.....	65
	Tool Number and Comment	68
	CRC Offset	70
	Spindle Direction	70
	Diameter	71
	Radius	72
	Tool Length	73
	4 Axis Safe	74
	Tool Type and Tool Category	75
	Vertical and Horizontal Offset	77

Cutter Bridge	78
Aggregate Offset	80
Spindle Offset.....	81
Collet Offset.....	86
Combination of Spindle AND Collet Offset.....	94
Cutter Compensation	97
6 Modifying a Tool.....	100
Temporary Editing	100
Permanent Editing	104
Moving an Item in the List	113
Changing Tool Parameters	115
7 Cutting Cycles.....	117
Cycle Descriptions	123
Profiling Cycles	127
Plunge-Outside	127
Plunge-Inside	132
Plunge-Outside Line Leads.....	137
Ramp-Outside	142
Ramp-Outside Partial.....	148
Ramp-Inside	154
Heli-Lead Outside.....	160
Heli-Lead-Inside.....	165
Heli-Lead Center.....	170
Multi-Pass-Outside.....	174
Multi-Pass-Inside.....	179
Incline-Outside	186
Incline-Inside	191
Incline-Outside-Ramp.....	196
Incline-Inside-Ramp.....	201
No-Retract-Outside.....	206
No-Retract-Inside.....	211
Taper-Plunge-Outside.....	216
Taper-Plunge-Inside.....	221
Profile-Inside-Auto	226
Profile-Inside-Interactive.....	231
Drilling and Hole Cutting Cycles.....	236
Drill Motions	236
Fast Drill	240
Drill-Interpolation.....	243
Spot-Bore	248
Peck-Bore	252
Pattern Recognition.....	256
Fast Pattern Recognition.....	260
Deep-Bore	264
Helical-Arcs-Center	267
Helical-Arcs-Edge.....	271
Hole-Interpolation 1.....	275
Hole-Interpolation 2.....	278
Pocketing Cycles.....	281
Linear-Pocket	281
Spiral-Pocket	285
Linear Pocket Ramp-In.....	289
Spiral Pocket Ramp-In.....	293

Linear Pocket Lead Hole.....	297
Advanced Rest Pocketing.....	301
Advanced Inward Pocketing.....	304
Advanced Outward Pocketing.....	307
Rectangular Fast Pocketing.....	310
Open Shape and Center Cutting Cycles.....	315
Center-Line Cut.....	315
Center-Line Ramp.....	320
3D Poly Center.....	325
Profile ZigZag.....	328
Incline-Center.....	332
Right Hand Cut.....	337
Left Hand Cut.....	341
NURBS Surface Machining Cycles.....	345
NURBS Cycles Supplied.....	346
NURBS Cycle Parameters.....	348
Knowledge Editor in NURBS.....	352
Tools in XNURBS.....	356
Establishing the Feed Plane.....	356
Z Level Completion Option.....	357
Fall over Mode in Vertical Cutting.....	359
Containment Cutting.....	362
Constant Scallop Height.....	365
Intersect Cutting.....	368
Roughing.....	373
Vertical Roughing Mode 1.....	375
Vertical Roughing Mode 2.....	378
Vertical Roughing Mode 3.....	381
Examples.....	384
NURBS XZ.....	384
Vertical Finishing at Angle 0.0.....	385
Vertical Finishing at Angle Minus 45.....	387
NURBS YZ.....	390
Vertical Finishing in YZ.....	391
NURBS CCW XY.....	394
Horizontal Finishing XY.....	395
NURBS Flow cut.....	398
Open Polyline Cross Flow Cutting.....	400
Closed Polyline Step Over Flow Cutting.....	404
Closed Polyline Spiral Flow Cutting.....	406
Closed Polyline Cross Flow Cutting.....	408
NURBS Project.....	411
Polyline Project Cutting.....	412
Specialty Cycles.....	415
4th Axis Machining.....	415
4 Axis Cutting.....	421
4 Axis Saw.....	425
4 Axis Moulder Head.....	429
DO-IT.....	433
Tap-Matic.....	434
Square Corners.....	437
T-Slot Plunge.....	442
Advanced Cycles.....	446
2D_Pocket.....	446

Advanced Spiral Pocketing.....	450
Engrave	453
Cycle Parameters	456
Offset Dim.....	456
Cut Side	457
Cut Direction.....	460
Round Corners.....	462
Lead-In	463
Lead-Out.....	469
Lead Size.....	475
Lead Angle.....	476
Lead Feed.....	477
Ramp Amt.....	478
Overlap Amt.....	478
Lead Ratio.....	479
MaxOffset.....	480
XY Step	481
XY Stock Allowance.....	482
Z Stock Allowance.....	482
LastOffset.....	483
Cornerpunch.....	484
Corner Clean.....	486
8 Status Information.....	488
Safety Plane	489
Depth Per Pass	490
Total Cut Depth	490
Material Thickness Compensation	491
Feedrate	492
Spindle Speed	492
Surface FPM and Units Per Rev.	492
Ramp Feedrate	494
Before Codes and After Codes	494
Oscillation Amt.	495
Sort by Rank #	497
Acc-N-Dec	497
Metric	500
Inline	500
Plane Detect	501
DOIT File	503
9 Cutting.....	503
Layer Control	503
Collect and Restore	504
Cutting Text (LetterEase)	509
Baseline Object.....	511
Edit	521
Justify	524
Spacing	531
Toolcheck	532
Erasecheck	533
Cutting Solids in Router-CIM	534
NCExtrude.....	535
NCExtract.....	539
NCDepth	541

	NCUpdate.....	546
	NCFlat	548
	Shapehole.....	550
	Parametric Toolpath in Router-CIM.....	551
	Cutting an Ellipse in Router-CIM	551
	Cutting a Spline in Router-CIM	558
10	Sequence and Make Code.....	558
	Sequence Definition	559
	Clear All	559
	Develop a Cut Sequence.....	561
	Sort Cuts By.....	562
	Tooling Requirements	575
	Make Subroutines.....	576
	Develop Tooling Motions	578
	Block Sequence.....	578
	Start and End Codes.....	579
	Information Development	581
	Make NC Code.....	581
	Make NC Code Options	582
	Acceleration/Deceleration.....	585
	Perform Machine Calc.....	589
	Report Information.....	591
	Edit Tooling Motions.....	591
	Sequence Emulator Options.....	593
	Sequence Undo	595
11	CutSIM in AutoCAD.....	595
12	Knowledge.....	596
	Retrieve Knowledge	596
	Save Knowledge	599
	Export Knowledge	601
	Import Knowledge	605
	Edit Knowledge	607
	Group Knowledge	614
	Save Without Knowledge	619
	Knowledge Notes	621
13	Tabbing.....	621
	Tabbing Settings	622
	2D or 3D Tabs	627
14	4th Axis Programming.....	629
	Geometry	629
	Cutting	631
	4th Axis Interpolation Cycle	634
	Draw ing Requirements.....	635
	Cycle Corner Treatments.....	635
	Outside Corners.....	636
	Inside Corners.....	636
	Position and Direction Control.....	637
	Lead Styles.....	638
15	Horizontal Boring.....	647
	Horizontal Boring	647
	Using FS, RS, BS, LS	648
	Setting up Knowledge	649

Using the Sample Drawing	657
Avoiding Extra Safety Moves	658
16 Router-CIM Drawing Tools.....	660
Creating a 'Slot' in AutoCAD	660
Creating a 'Bubble Fillet' in AutoCAD	662
Converting 2D to 3D Polyline	664

Part III Router-CIM Automation Suite 666

1 Starting Router-CIM Automation Suite	666
2 The Automation Interface.....	667
Menu Area	667
File	668
Database Maintenance	669
Backup	670
Restore Over	672
Restore New	674
Set as Current	676
Delete	677
New (Database).....	678
Verify Server	679
Misc. Utilities	680
Shared Databases.....	681
Installing on a Stand-Alone Server	685
Settings	686
General Settings	686
ShapeDone Settings	692
System Folder Settings.....	693
Layer List Settings.....	697
Field Caption Settings.....	698
Import Wizard Settings.....	701
Label Generation Settings.....	703
Tool Tip Settings.....	703
Command Center.....	704
Advanced Settings	705
Nesting Engine Settings	707
Print Options	708
Exit	708
Tools	709
Create New Knowledge Drawing.....	709
Create Pack and Go.....	709
Clear AutoNest Task and Part Folders.....	711
Clear ShapeDone Folder.....	711
Restart AutoCAD UI Engine.....	712
Restore Startup Drawing.....	713
Save Current Settings as Job Default Settings.....	714
Automatic G-Code Scan.....	714
Migration Wizard.....	717
Multi Post Default.....	717
NCVAR Editor	717
Label Designer	729
Single Part Report Configuration.....	730
Import Wizards.....	731
Excel and Comma Delimited.....	732

Solid-CIM CSV	739
Pack and Go	740
Select and Go	741
Additional Imports.....	741
CabinetVision	742
Cabnetware	744
KCDw	748
Excel and Comma Delimited Version 1.....	749
Pattern Systems.....	757
View	758
Help	759
Toolbar	760
New Folder.....	760
Job Wizard.....	761
New Job	767
Add Part	767
MPR To DWG Converter.....	768
Add Assembly.....	769
Part Properties.....	771
Edit Geometry.....	772
Delete Folder/Job.....	772
Delete Part.....	773
Cad System.....	773
Material Database.....	773
Results Folder.....	774
Run Job	774
Folder Tree	774
Batch Jobs.....	777
Right Click Options.....	780
Drag and Drop Job/Folder.....	783
Part Window	785
Part Properties.....	787
Job Settings	793
Nesting	799
Advanced Nesting.....	808
Nesting Bridge Parameters	809
Automated Two-Side Nesting.....	818
Cart Control Nesting.....	828
Veneer Matching.....	837
Small Part Zones.....	844
Staydown and PIP Bridging.....	848
Skeleton Cutting Control.....	850
Dynamic Variables.....	852
Printing and Labeling.....	852
Part Name.....	866
Output Files.....	868
Results	870
Part Window Right Click Options.....	871
Drag and Drop Parts.....	873
Variables Window	874
Part Preview Window	876
3 Moving Windows in Automation.....	877
4 Knowledge Settings.....	878

Edit Knowledge Drawing	881
Edit DOIT File	883
5 Multi Post Processing.....	902
6 The Results Folder	909
AnestParts	910
AnestTasks	911
Code Single DWGs	912
Code Single Out	913
Labels	914
Misc	915
NestedDWGs	916
Simulation Files	917
Summary Report	918
7 Material Database/Properties.....	919
Material Properties	920
Scrap Management	924
Edit Material	928
New	929
Duplicate Material	930
Delete	930
Import	931
Export	940
Delete Scrap	943
Delete Alternate Sizes	944
Materials with Scrap	944
Materials with Alternate Sizes	945
Materials in Use	945
Materials Not in Use	946
All Materials	946
8 Advanced Nesting Material.....	947
9 Labeling.....	953
Label Designer	953
Creating a Sheet Layout.....	955
Creating a Label.....	959
Define Label Outputs.....	976
Automation Label Settings.....	979
10 Advanced Topics.....	981
Pocketing in Automation	981
Alternating Tables on a Twin Table Machine	981
Cutting files from a Spreadsheet in Router-CIM Automation	982
Using PREDRILL Feature	986
Part IV Kinematic Time Study	988
1 Post Word Values.....	990
2 Machine Parameters.....	993
3 Tool Change Times.....	993
4 Maximum Machine Angle	994
Part V NCVARS	996

1	NCVARs Appendix.....	996
	Common NCVARs in Automation	996
	Geometric	1000
	Layer	1000
	System	1001
	Other	1003

Part VI AutoNest Manual 1005

1	Getting Started With AutoNest.....	1006
	AutoCAD Environment	1006
	Command Flow	1007
	Command Overview	1009
	Sample Nests	1011
2	Commands.....	1012
	Sysdata	1012
	Anest Parts	1021
	Save Part.....	1022
	Import Rectangle Part.....	1024
	Convert Part.....	1028
	View Part.....	1031
	Save Cluster Part.....	1032
	View Cluster Part.....	1035
	Bridge Part (Staydown).....	1036
	Break Part.....	1038
	Mirror Part.....	1042
	Expand Part.....	1044
	Anest Irregular Stock	1045
	Save IR Stock.....	1046
	Convert IR Stock.....	1048
	View IR Stock.....	1049
	Anest Task Edit	1050
	Anest Nesting	1076
	AutoNest Nesting Single Task.....	1077
	AutoNest Batch Nesting.....	1082
	Anest View Nest	1089
	Anest View Report	1093
	Anest Update Nest	1096
	Anest On-Line Help	1096
	Anest License	1098
	About AutoNest	1101
3	File Format.....	1101
	.VEC File Format	1102
	.STK File Format	1105
	.CLS File Format	1107
	.JOB File Format	1109
	.SYM File Format	1116
	.SUM File Format	1121
	.SGD File Format	1122
	.XML File Format	1125
	.DXF File Format	1127
4	Error Messages.....	1131
	Reference To Error Messages	1132

5 System Parameters.....	1138
Format for ANEST.SYS	1138
Format for ANEST.TOL	1146
Format for ANEST.SET	1148

Part VII Advanced Programmer Topics 1160

1 Post Processor.....	1160
Post Processor Considerations	1161
Post Processor Output	1162
Using the Router-CIM Post Processor	1163
Cut and Sequence	1163
Processing Concepts	1164
Connecting the Dots	1165
Configuring the "\$PP" File	1166
Translation of text statements from a Drawing	1167
Format of AutoCAD Text Statements	1167
Translation of Geometry from a Drawing	1168
Motion Block Definition	1168
Group Definition and 'G' Switch	1169
Absolute Preset Example	1170
Pocketing Example	1171
Switches	1172
Numeric Formatting	1173
Global Numeric Formatting	1173
Individual Numeric Formatting	1174
Modals	1174
ASCII Modals	1175
NUMERIC Modals	1176
AXIS Modals	1176
Reserved Words	1177
Fine Tuning	1186
Switch Parameters	1186
General Format of the ;P Switch	1189
"F" Switch Parameters	1190
Geometric Entries	1192
Customization	1194
2 Video Jet print head.....	1195
Video Jet FAQ	1197
Komo Production Manager and Video Jet	1198
Inkjet M-Codes	1199
3 How to Backplot NC Code.....	1200

Part VIII Parametric Macro Builder 1207

1 Macro Builder Interface.....	1208
File Menus	1210
Explorer	1211
Explorer Interface.....	1211
Explorer Menu Options.....	1213
Viewer	1215
View er Interface.....	1215
Execute Draw ing & 3D Solid.....	1217
Editor	1219

Editor Interface.....	1220
Defaults & Validation.....	1223
Sides, Corners & UCS.....	1224
2 Operations.....	1226
CutOut	1227
Dado/Saw	1231
Drill	1232
Global Variables	1236
Local & Mac Variables	1238
Panel	1247
Profile	1248
Text	1251
ToeKick	1252
3 Programming Reference.....	1253
4 Evaluation Error.....	1255
 Index	 1257

Router-CIM Automation Suite 2025 Minimum System Requirements

For 64-bit Systems

- Microsoft® Windows® 11 (64-bit) or Microsoft® Windows® 10 (64-bit) version 1809 or above.
- 2.5-2.9 gigahertz (GHz) or greater (Recommended: 3.0+ GHz or greater, 4 or more cores). ARM Processors are not supported.
- 8GB of RAM (32GB or more for 3D Modeling)
- 10GB or more Free disk space for Installation (40GB or more for 3D Modeling). SSD suggested.
- 1920 x 1080 True Color video display adapter 128 MB or greater, Pixel Shader 3.0 or greater, with "Recommended" graphics card (3840 x 2160 (4K) with preferred scaling: 100%, 125%, 150% or 200% on Windows®, 64-bit systems with capable display card)
- Microsoft® Internet Explorer 11.0 (or equivalent) or later web browser with network connection
- 2 GB GPU with 29 GB/s Bandwidth and DirectX 11 compliant (Recommended: 8GB GPU with 106 GB/S Bandwidth and DirectX 12 compliant)
- .NET Framework Version 8 or later
- 1 Available USB port

Supported AutoCAD Versions

Router-CIM 2025 supports the following Autodesk products:
AutoCAD 2025

You should check your hardware against the Autodesk Certified Hardware List to be sure of compatibility with your version of AutoCAD.

Hardware Locks

In addition to the hardware requirements listed above, it is important to note that Router-CIM and AutoNEST require Hardware Locks (often called 'dongles') in order to function. Since there are two Hardware Locks required if you are using the nest software, you should have two USB ports available.

Note on Hardware Locks:

The Router-CIM 2007-2025 Hardware Locks are different than the 2004, 2005, and 2006 Hardware Locks. If you are upgrading from a version prior to 2007, you will receive a new Hardware Lock with the software and will have to return the old Hardware Lock through a traceable carrier. Lost locks are the customer's responsibility so insure your shipment for full software value. If you fail to return the old Hardware Lock, you will be invoiced for the full purchase price of the software. If you have Router-CIM 2007 or newer, the Hardware Locks are compatible and no switch is necessary.

Note: Type 1 Hypervisor Appliance Compatibility Warning: Please be aware that neither Router-CIM, Solid-CIM or Radan's Autonest USB hardware locks are compatible with any Type 1 Hypervisor appliance, including, but not limited to, Microsoft Hyper-V, Oracle VM, KVM and Citrix Hypervisor. These products ARE compatible with Type 2 Hypervisor appliances, such as VMWare's VM Workstation.

If you are using parallel port Hardware Locks instead of USB Hardware Locks, it is not suggested to have a printer, ZIP disk, scanner or any other device connected to the same port as your AutoNest Hardware Lock. Certain devices may cause Hardware Lock failure, which will render the software useless until the Hardware Lock is replaced. Other printing conflicts may occur while printing through the same port as the Hardware Lock. It may be necessary to have a second parallel port dedicated specifically to the Hardware Lock.

Product Overview

The main purpose of Router-CIM Automation Suite is to provide a complete CAD/CAM solution for both single part machining and nested based manufacturing. Router-CIM Automation Suite allows you to automatically produce NC Code for parts that are produced or imported into the AutoCAD environment, as well as any other single part that can be parametrically defined, or produced as a drawing or DXF file.

For new users of Router-CIM Automation Suite, there are a few concepts which need to be understood in order to use the product efficiently. These are:

- A basic understanding of the Microsoft Windows environment, and file handling with programs such as My Computer, or Windows Explorer.
- A working knowledge of the AutoCAD environment. How to create and name layers, and assign geometry to those layers.
- Some knowledge of the types of tools, cutting conditions, and materials you are likely to use on your parts.
- Knowing the difference between the file types Router-CIM Automation Suite uses such as DWG, DXF, and SCN.
- Understanding the Layer to Knowledge association feature in Router-CIM Automation Suite.

If you will be primarily using macros for your part definitions, then a complete understanding of Router-CIM's Parametric Macro Builder and how to create a macro is truly necessary. This takes practice and is best done on paper first! Also you will need to understand the different variable types: Global, Dynamic, Tagged, and Local, and how to use them.

Installation Quick Start

These installation notes assume that AutoCAD 2025 is installed and functioning properly.

Make sure you are logged in with at least Local Administrator privileges on your system.

The Router-CIM program is installed in the following manner.

- Unplug any USB hardware locks previously installed.
- Download the Router-CIM installation files according to the included download card with your Router-CIM software package or visit www.cim-tech.com/20RCIM25. The website will include the instructions for downloading the necessary files including the file for starting the installation.
- The Router-CIM serial number will determine the installation options. The serial number is located on the Router-CIM download card included with your Router-CIM software package. If Nesting was purchased, the Router-CIM installation will automatically install the product.
- The Install will place the Router-CIM files in the default locations.
- The install procedure searches for AutoCAD on your hard disk, and uses this location to build icons to run the Router-CIM program.
- Once the install is finished, reboot the computer, plug in the USB hardware locks and you will be able to run the product.
- If any custom post processors are necessary, install them now. The procedure to install the post processors is to simply run the .exe file that contains the post processor, and answer a few default questions during the install.

Critical Issues



After Router-CIM is properly installed and configured for use, all pertinent folders should be backed up. A simple method is by copying them and placing them in a new folder. The proper way would be to make a CD, external hard drive or another easy to restore method.

CD Recorders, DVD Recorders and External Hard Drives are all valid ways to back these folders up. Typically all of the folders can fit on one CD. After using the system for a while, the backups that Router-CIM stores on your system could mean that more than one CD is necessary.

A Note on CD Recorders: Each file that is backed up in this fashion may become read-only. This means you can only look at the documents without altering or editing them. It will be necessary to change the properties of the files back from read-only after the folder is copied. The method of doing this varies depending on your operating system.

The folders to back up are as follows:

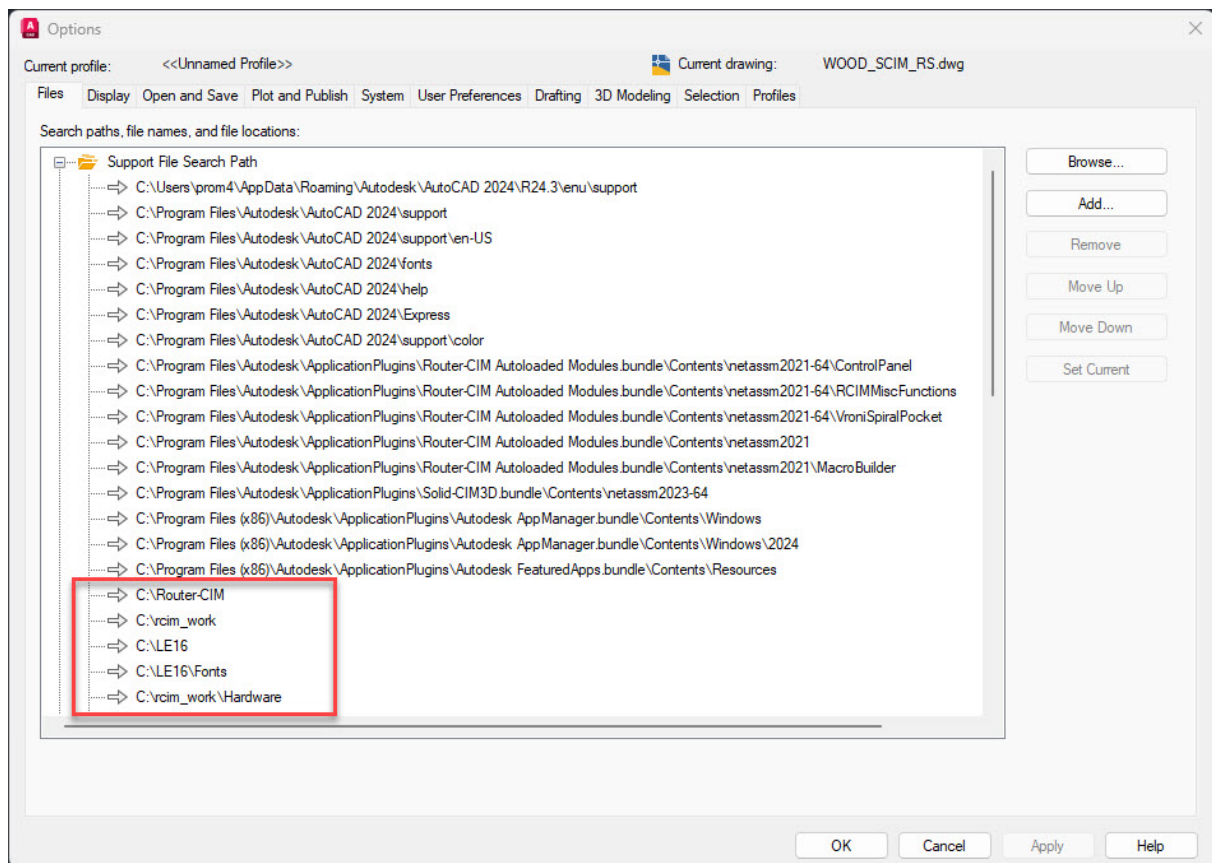
1. **C:\Router-CIM**
2. **C:\Rcim_work**
3. **C:\Automation_code**

(C: is assumed here to be the installation drive.)

CIM-Tech is NOT Responsible for your loss of productivity or data! If you do not back up your system completely and you lose data without a backup, your only option is to re-install the software and start over.

Once Router-CIM has been installed, you can verify that the correct AutoCAD Support File Search Paths have been added by opening AutoCAD and typing OPTIONS in the command line.

Under the 'Files' tab, you can verify the five folder locations as highlighted below.



Installation Guide

Notes:

- 1) Local users will need Administrator privileges to install the software.**
- 2) You should remove any Hardware Locks that have been plugged into the computer.**
- 3) Any Anti-Virus or Malware Software should be disabled temporarily prior to starting the installation. Make sure to enable these when the installation has been completed**
- 4) These installation notes assume that AutoCAD 2025 is installed and functioning properly.**

Failure to do these things may result in an incomplete installation causing the program to not function properly.

Note: If you need to install Router-CIM on a different computer and want to transfer your information, please read the 'Installing on a Different Computer' section of the installation manual.

The Router-CIM program is installed in the following manner:

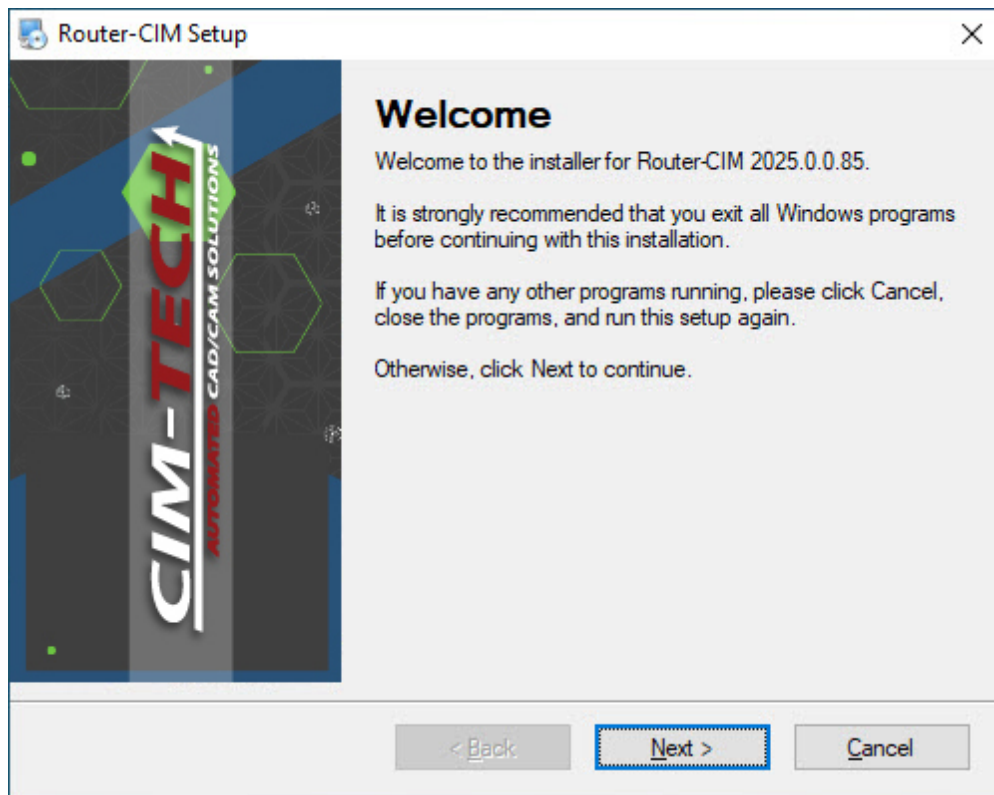
Download the Router-CIM installation files according to the included download card with your Router-CIM software package or visit www.cim-tech.com/20RCIM25. The website will include the instructions for downloading the necessary files including the file for starting the installation.

The Router-CIM serial number will determine the installation options. The serial number is located on the Router-CIM download card included with your Router-CIM software package.

The install will search for Router-CIM files in the default location.

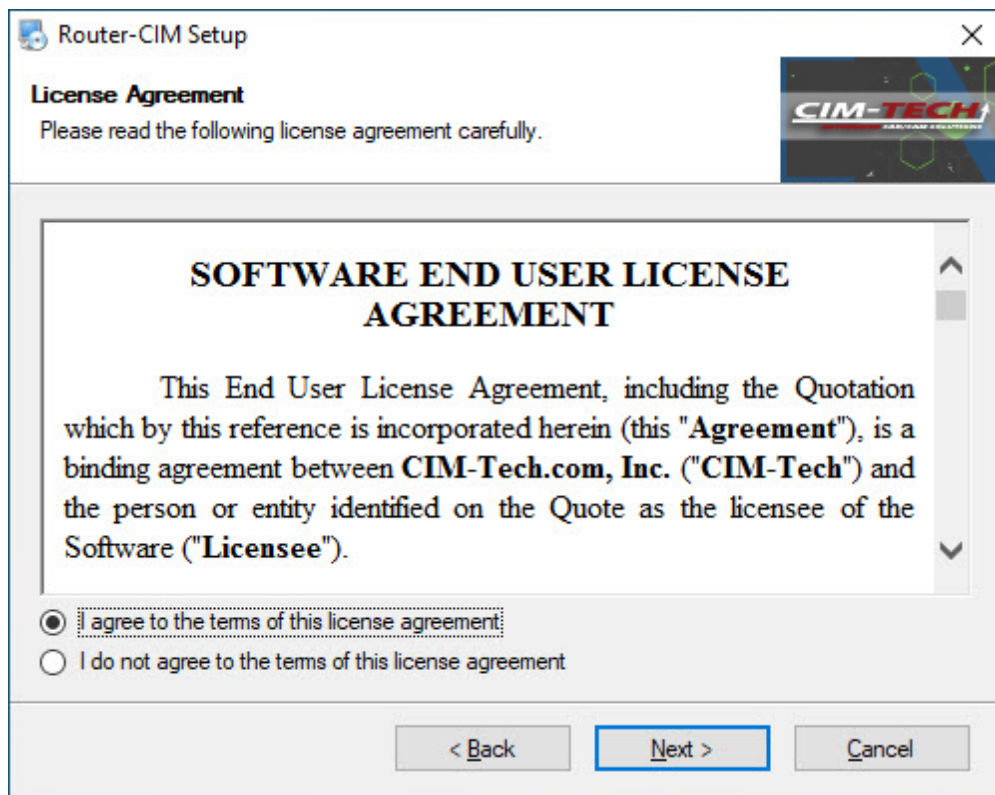
The install procedure searches for AutoCAD on your hard drive, and uses this location to build icons to run the Router-CIM program.

When the installation program starts, the following screen will appear:

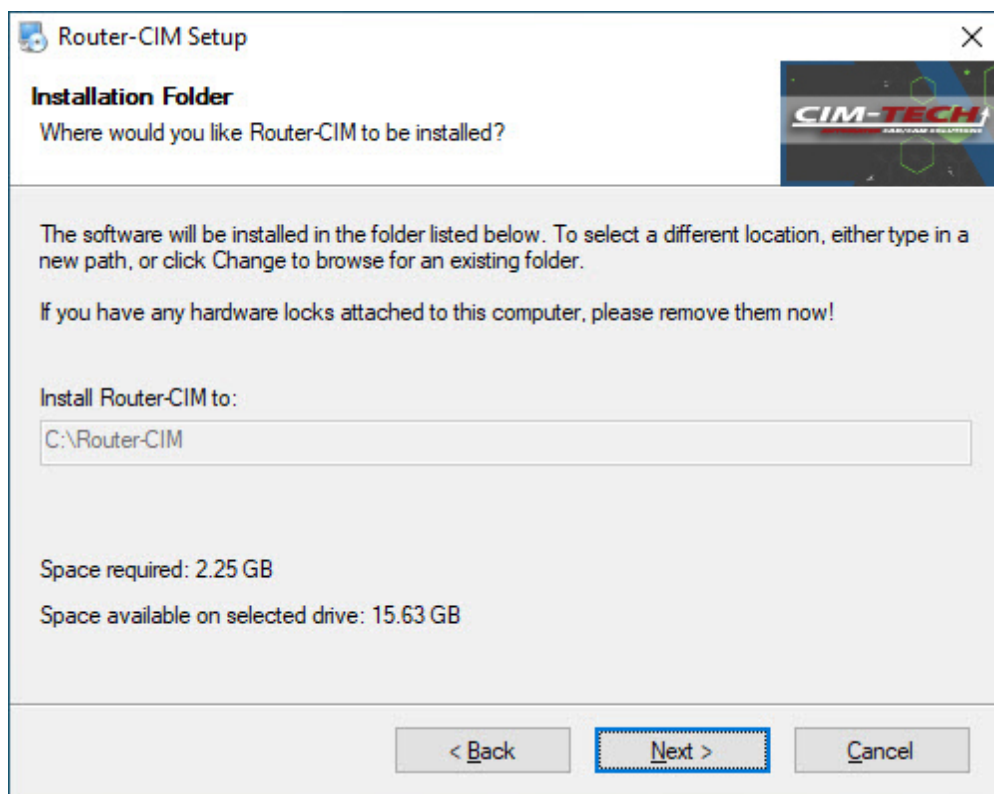


From here, all of the default selections have been made for you.

Select '**Next**' to continue, or '**Cancel**' to exit from the installation.

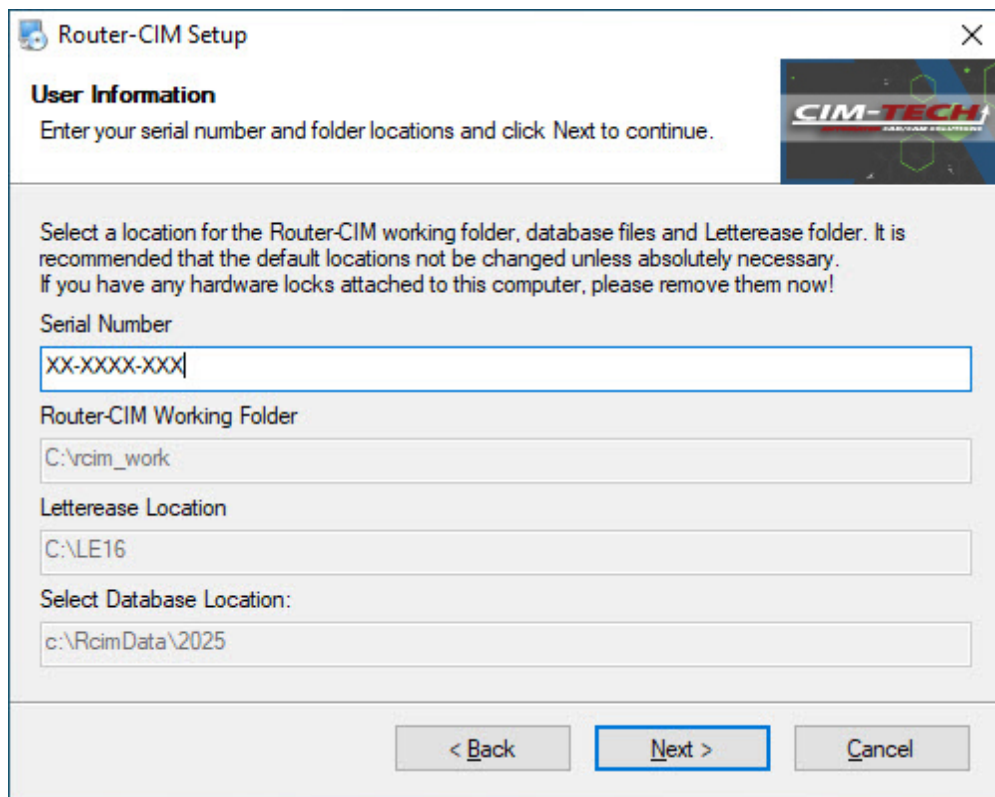


This is the license agreement for Router-CIM. Be sure you read the agreement. Selecting '**I do not agree to the terms of this license agreement**' will allow you to exit the program without installing it to your computer. Selecting '**Back**' will return you to the previous window. Select '**I agree to the terms of this license agreement**' and '**Next**' to continue.



Router-CIM will specify the location for installation. You may select **'Next'** to continue, **'Back'** to go to the previous window, or **'Cancel'** to exit from the installation.

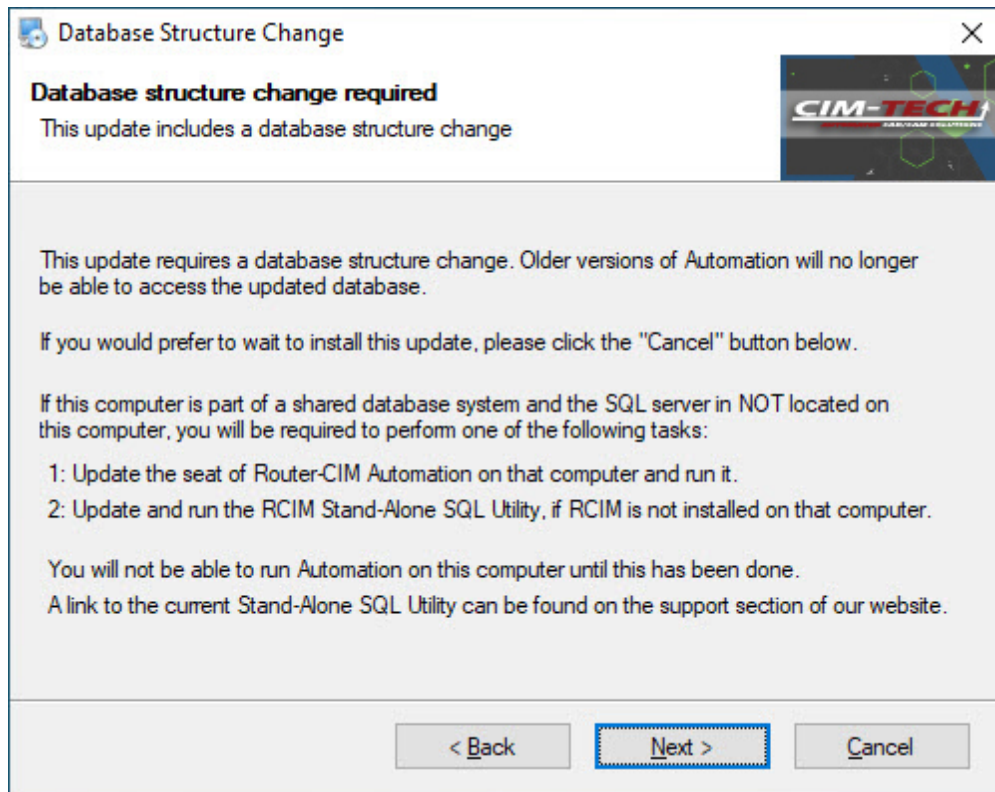
You should remove any hardware locks that have been plugged into the computer.



The image shows a screenshot of the 'Router-CIM Setup' window. The title bar says 'Router-CIM Setup' with a close button. The window has a 'User Information' section with the instruction: 'Enter your serial number and folder locations and click Next to continue.' Below this, there is a text box for the 'Serial Number' with the placeholder 'XX-XXXX-XXX'. There are three more text boxes for folder locations: 'Router-CIM Working Folder' (containing 'C:\rcim_work'), 'Letterease Location' (containing 'C:\LE16'), and 'Select Database Location:' (containing 'c:\RcimData\2025'). At the bottom, there are three buttons: '< Back', 'Next >' (which is highlighted with a blue border), and 'Cancel'. In the top right corner, there is a small graphic with the text 'CIM-TECH' and 'ASSOCIATED EQUIPMENT'.

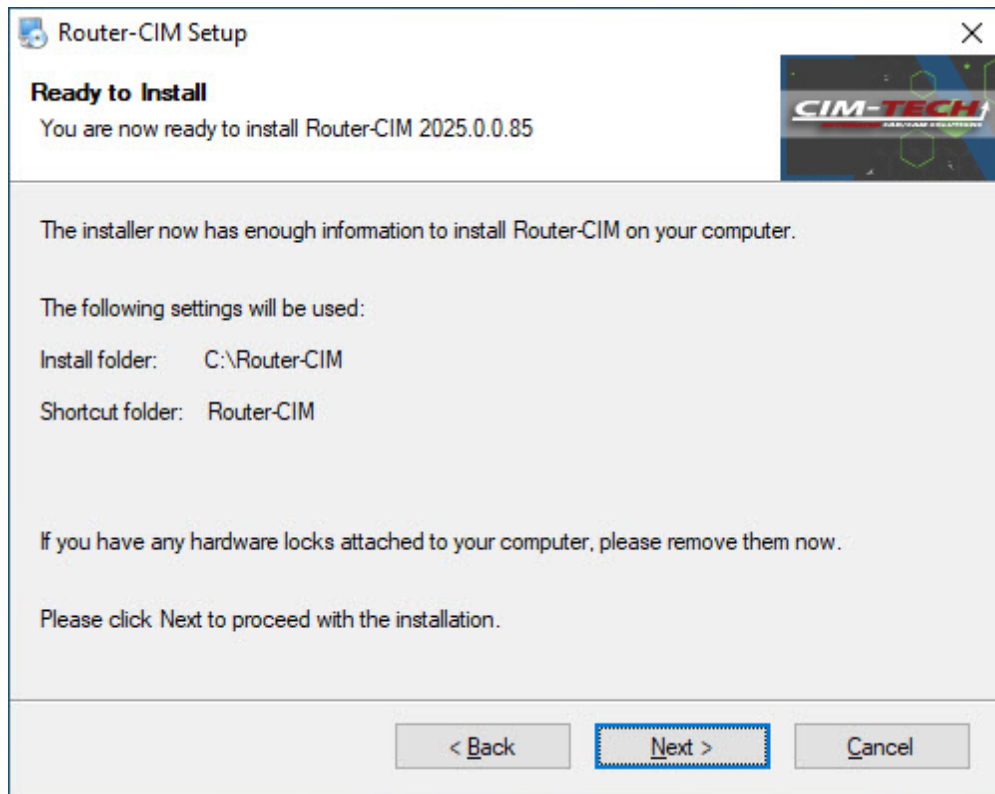
Enter your serial number that is supplied to you with the software. The default database location is on your C: drive in the RcimData folder.

You may select **'Next'** to continue, **'Back'** to go to the previous window, or **'Cancel'** to exit from the installation.



This notification screen will appear informing you that the SQL Database will be updated upon completion of the installation. If you are running a shared database between multiple Router-CIM users, you will need to make sure all users accessing the database are updated to the same version.

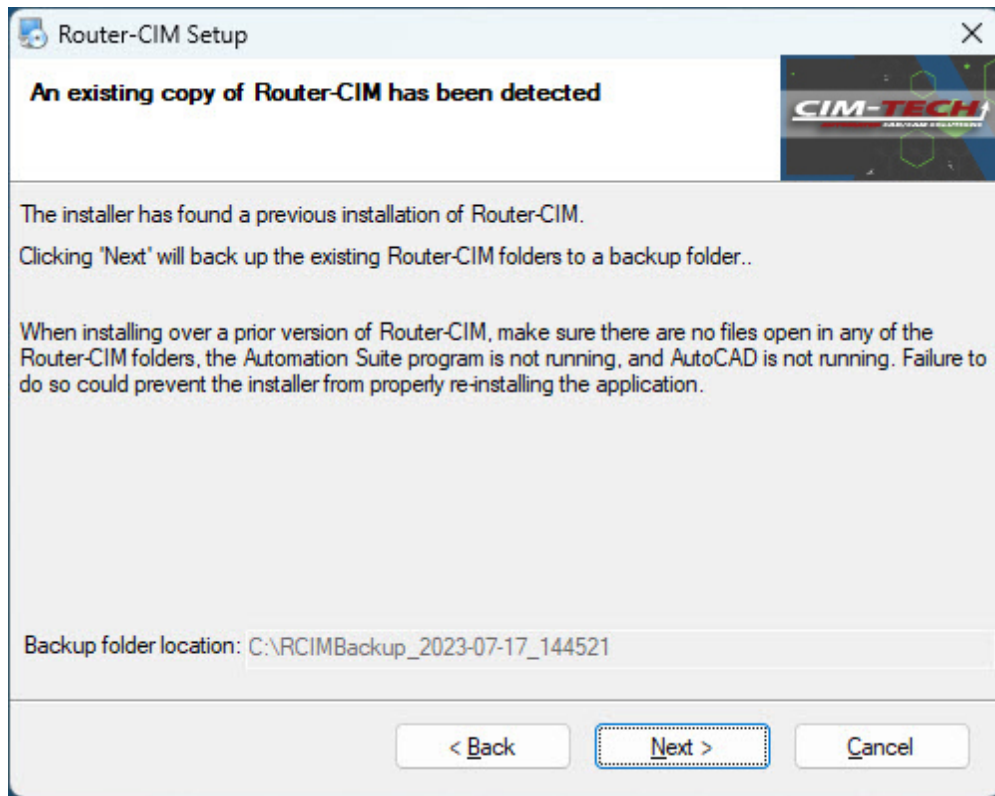
You may select **'Next'** to continue, **'Back'** to go to the previous window, or **'Cancel'** to exit from the installation.



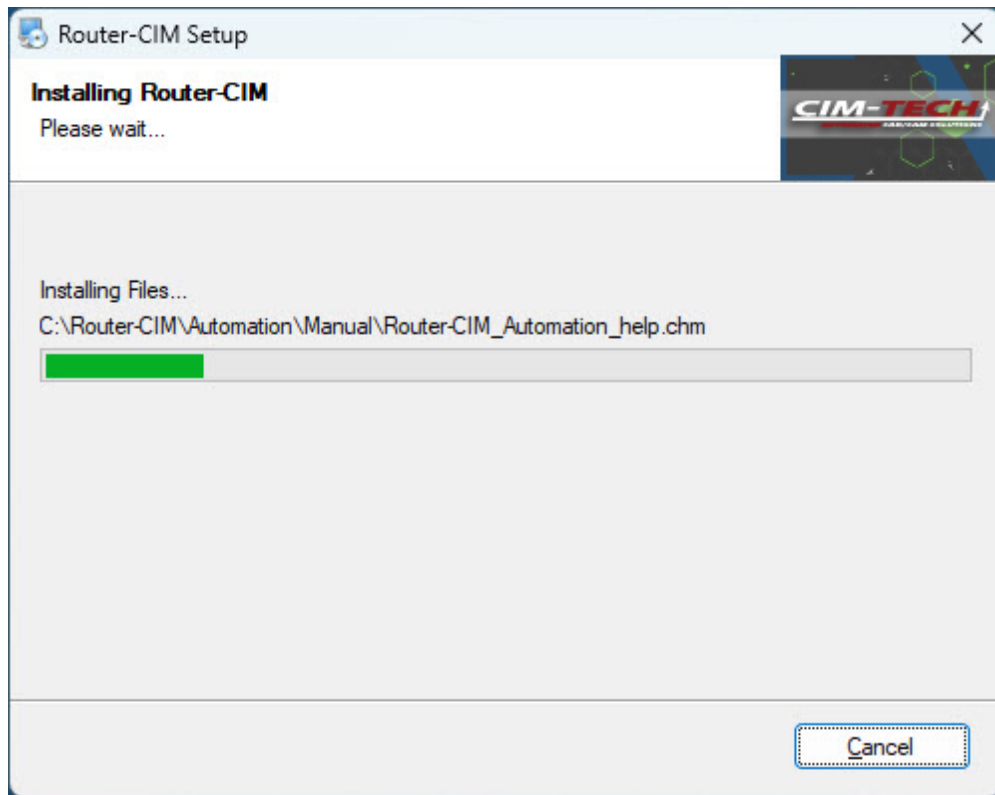
This screen will recap where Router-CIM is going to be installed to.

You may select **'Next'** to continue, **'Back'** to go to the previous window, or **'Cancel'** to exit from the installation.

Note: If an existing copy of Router-CIM is found on the computer in which you are installing, you will see this screen:



You may select **'Next'** to continue, **'Back'** to go to the previous window, or **'Cancel'** to exit from the installation.

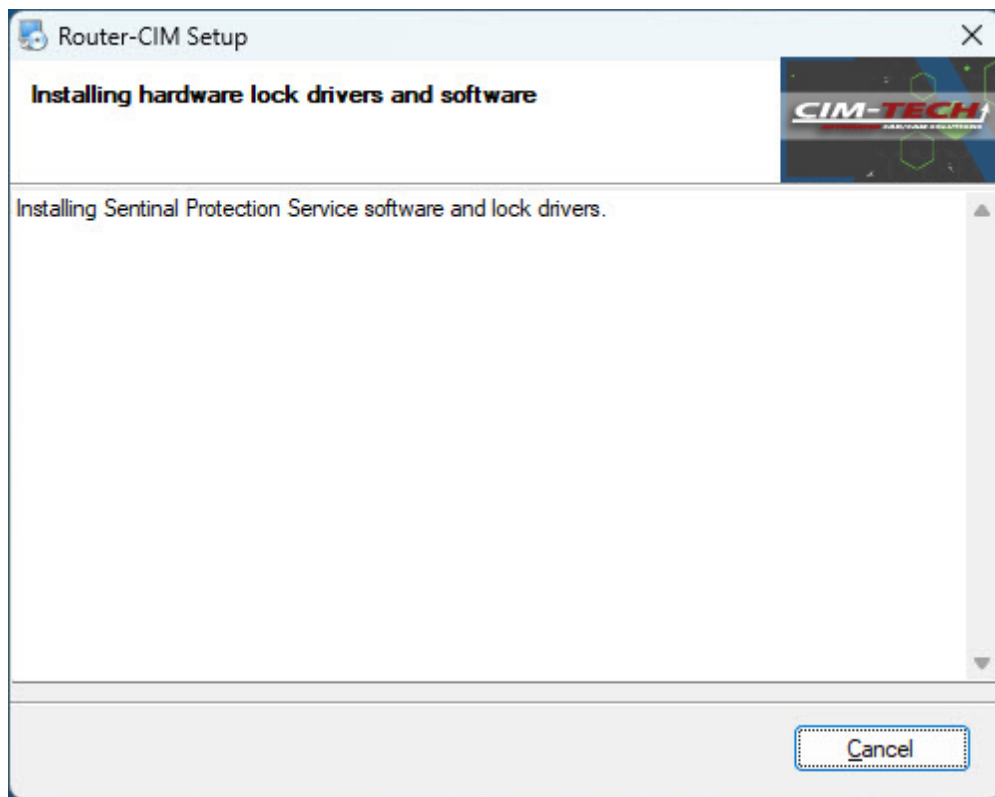


The installer will start copying files to the necessary locations in this section of the install.

The SQL Server is the second section of the installation. The install will extract some more files and then check your computer for versions of AutoCAD, SQL Server and Microsoft .NET Framework versions.

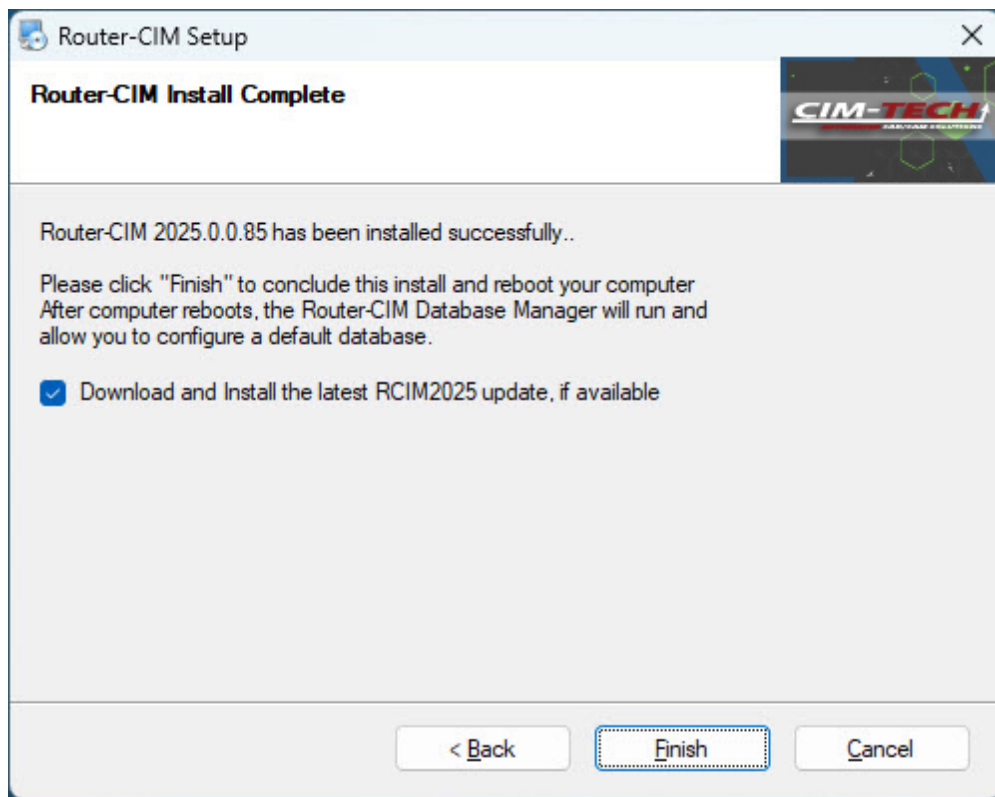
This section may take a while, please be patient and allow the installer to finish.

The next step will appear when this is finished.



The Sentinel Protection Installer will run next to install the hardware lock drivers. This is usually pretty quick and you may see some windows open with a black background while this is installing.

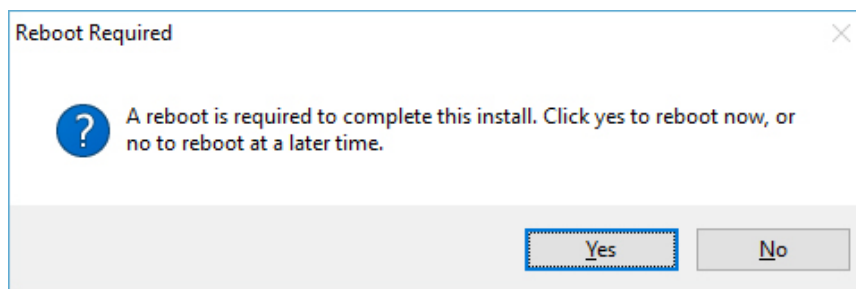
Once that is finished, the install of Router-CIM is completed and the following screen will appear:



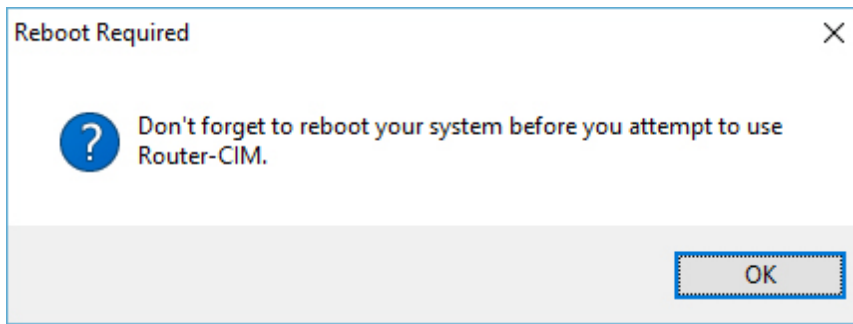
Note: After the Router-CIM Installation, Router-CIM will check to see if an update is released if the option for 'Download and Install the latest RCIM update, if available' is checked.

Select '**Finish**'.

You will see a message requesting you to reboot your computer. The installation is not finished until you reboot your computer.



If you select '**No**' at this time, a reminder screen will appear.



Once a reboot has been completed, the installation of Router-CIM is complete. There is a USB lock for a USB port that will need to be plugged in at this time.

Installing On A Different Computer

Note: Follow the procedure for installing Router-CIM described under the 'Installation Guide' section and then proceed with this section.

- 1) Install your Post Processor(s) from the same setup files you used on the previous system. Or, create a pack and go on the previous system and import it on the new system.
- 2) Install any optional add-ons such as Advanced Nesting, CutSim, Multi-CIM or Solid-CIM 3D. If you are installing Solid-CIM 3D, be sure to copy your authorization file (rcimauth.key) to C:\Router-CIM. This file would have been emailed to you at the time the lock was authorized. You can also copy it from the C:\Router-CIM folder of a previous install or backup.
- 3) Knowledge drawings and DOIT files are a simple copy and paste. Their location is set from Router-CIM Automation Suite under File > Settings > System Folders . Find these files on the old system and place them in the same folder structure on the new system. If your files are located on a network that the new computer also has access to, you will not need to move the files.
- 4) Your materials, job library, and Automation settings are stored in the database.
 - A) Either click **File > Database Maintenance** or run the **RCIM DB Manager** shortcut to access the database utilities window.
 - B) From the previous system, use the **'Backup'** button to create a backup of the database(s).
 - C) Copy the backup file from C:\Router-CIM\Automation\Database\Backups to the same location on the new system.

Note: The backup(s) must be in that folder to be able to restore them.

D) On the new system, use the '[Restore New](#)' button in the [RCIM DB Manager](#) to restore the database backup(s).

Updates

The Router-CIM installs a complete version of Router-CIM. There are, however, updates made to the product on a regular basis.

Please visit <http://www.cim-tech.com> for updates to Router-CIM. Go to the Support section of the website and select on 'Software Updates'.

Visit <http://www.cim-tech.com> for regular updates and information relating to the Router-CIM software.

Note: Router-CIM Automation Suite will also check regularly to see if a new update is available. To ensure that Router-CIM Automation Suite can access the updates, you will need to have a network connection and you will need to make sure that in Router-CIM Automation Suite your 'Settings' under the 'General Settings' tab has '[Check for Available Updates](#)' selected.

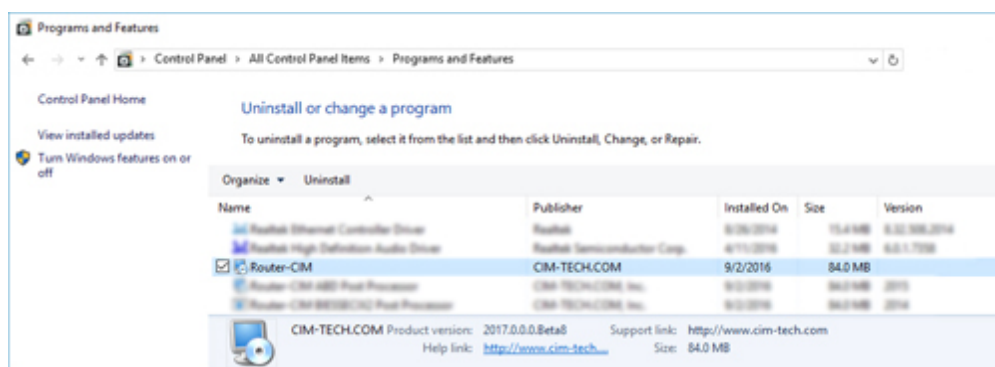
Uninstall Router-CIM

In order to uninstall Router-CIM, you should use the Add/Remove programs application in Windows.

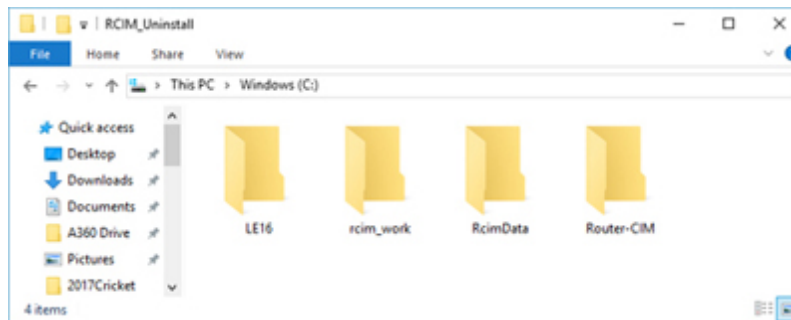
You will see the entry for **Router-CIM** in this section. You should remove the Router-CIM entry.

If you are not re-installing on the current machine, you can also remove the Sentinel Protection Installer x.x.x.x.

Note: If you have other software that uses a Sentinel hardware lock, you should not remove the Sentinel Protection Installer program. x.x.x will be identified by a specific number version dependent on which version was installed.



Once Router-CIM is uninstalled, you may remove the folders that remain on the C: drive that uninstall did not remove. Any folders which contain files that have been added or modified after the install will not be removed. The folders that can be removed are shown below:



Router-CIM

The main purpose of Router-CIM is to provide a complete CAD/CAM solution for both single part machining and nested based manufacturing.

For new users of Router-CIM, there are a few concepts which need to be understood in order to use the product efficiently. These are:

- A basic understanding of the Microsoft Windows environment, and file handling with programs such as My Computer, or Windows Explorer.
- A working knowledge of the AutoCAD environment. How to create and name layers, and assign geometry to those layers.
- Some knowledge of the types of tools, cutting conditions, and materials you are likely to use on your parts.
- Knowing the difference between the file types Router-CIM Automation Suite uses such as DWG, DXF, and SCN.

The following sections are broken up into the different areas of the Router-CIM program and are designed to give you as much information about the options available as possible.

This section is meant to be a reference for Router-CIM programming software and not a tutorial.

Router-CIM Interactive - Start Router-CIM in AutoCAD

To program with Router-CIM within an AutoCAD drawing, you need to start Router-CIM within the drawing.

First, open a new session or the drawing file in AutoCAD that you would like to program.

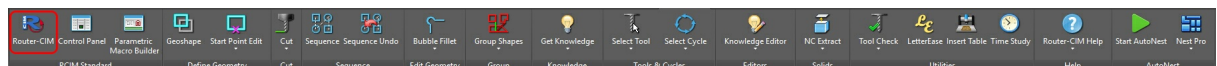
To start Router-CIM within an AutoCAD drawing, switch to the Router-CIM Ribbon bar or use the Router-CIM Toolbar:

Note: If you want to use the toolbar and you do not see it in the current version of AutoCAD, go to the [Router-CIM Toolbar](#) section.

Ribbon Bar:



Then select the far left icon in the ribbon bar:
























Or select the far left icon on the Toolbar:



Once you select the Router-CIM Icon, this will open up the '[Configuration Wizard](#)'. Go to the '[Configuration Wizard](#)' section for more information.

If Router-CIM was already started in the current drawing session, this will open up the 'Control Panel'.

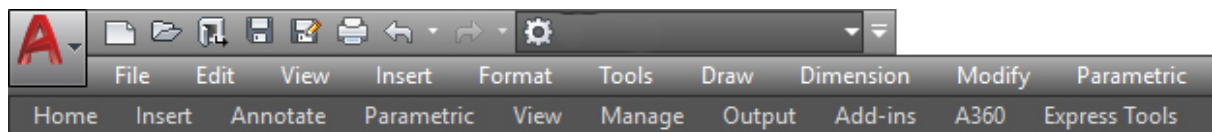
Here is a Quick Reference for the Router-CIM icons and keyboard shortcuts:

Function	Keyboard Shortcut	Icon	Function	Keyboard Shortcut	Icon	Function	Keyboard Shortcut	Icon
Control Panel	AS		Modify Tool	MT		Knowledge Editor	KE	
<u>GeoShape</u>	GS		Select Cycle	SC		DOIT Editor	DE	
Start Point Edit	SP		Modify Cycle	MC		Edit NC Code	EC	
Cut	CUT		Get Knowledge	K		<u>LetterEase</u>	LE	
Sequence	SE		Save Knowledge	SK		Tool Check	TC	
Sequence Undo	SU		Import Knowledge	IK		Group Shapes	GP	
Select Tool	PT		Export Knowledge	EK		Bubble Fillet	BF	

How to access the Router-CIM Toolbar in AutoCAD

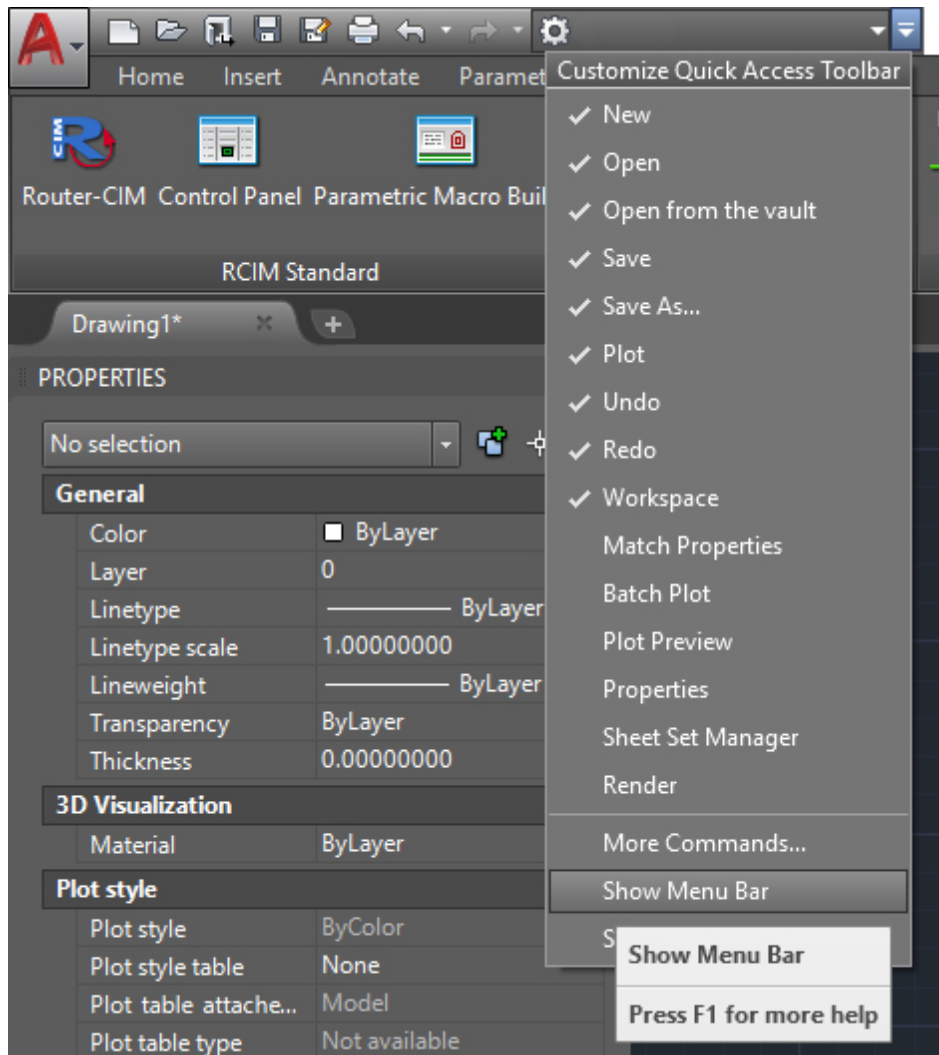
To access the Router-CIM Toolbar in AutoCAD, follow these steps:

Once AutoCAD is open, go to the AutoCAD Menubar:

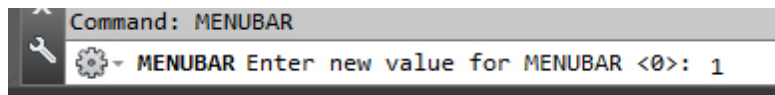


The Menubar is File - Edit - View and so on located at the top of the AutoCAD window.

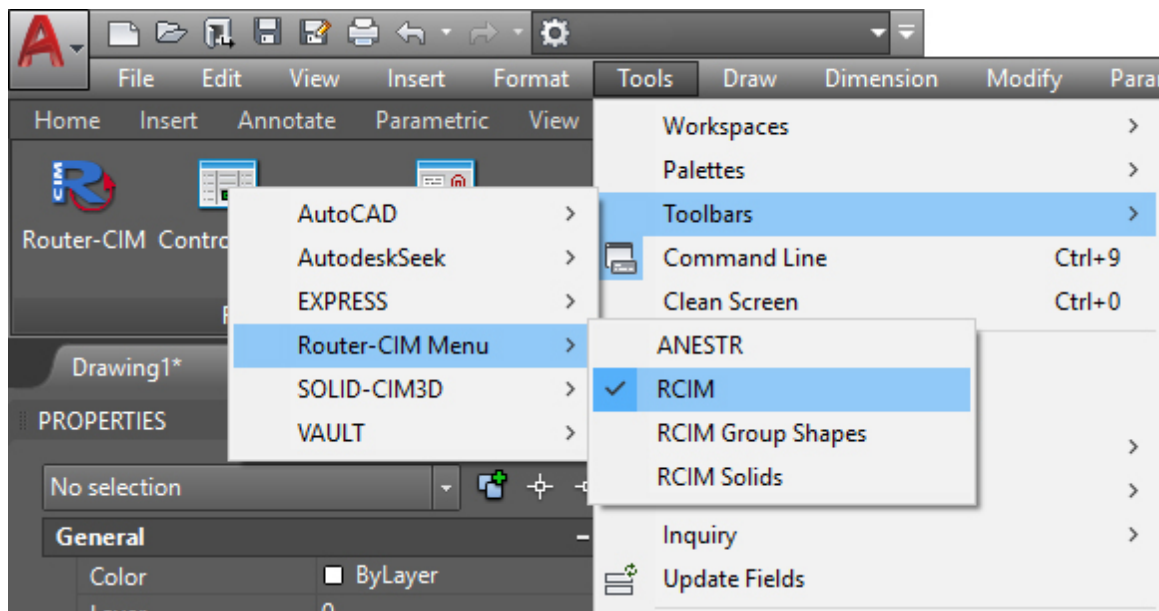
If you do not see the Menubar, go to the drop-down arrow beside your workspace settings and select the 'Show Menubar' option. You can also type Menubar into the AutoCAD command line and change the setting to 1.



Or:



Once the Menubar is visible, select the 'Tools' option, then 'Toolbars'. In the flyout menu, select 'Router-CIM Menu' and then the 'RCIM' option.



This will open the Router-CIM toolbar:



At this time you should repeat the process for the RCIM Group Shapes, RCIM Solids and ANESTR toolbars as well.

RCIM Group Shapes Toolbar:



RCIM Solids Toolbar:



ANESTR Toolbar:

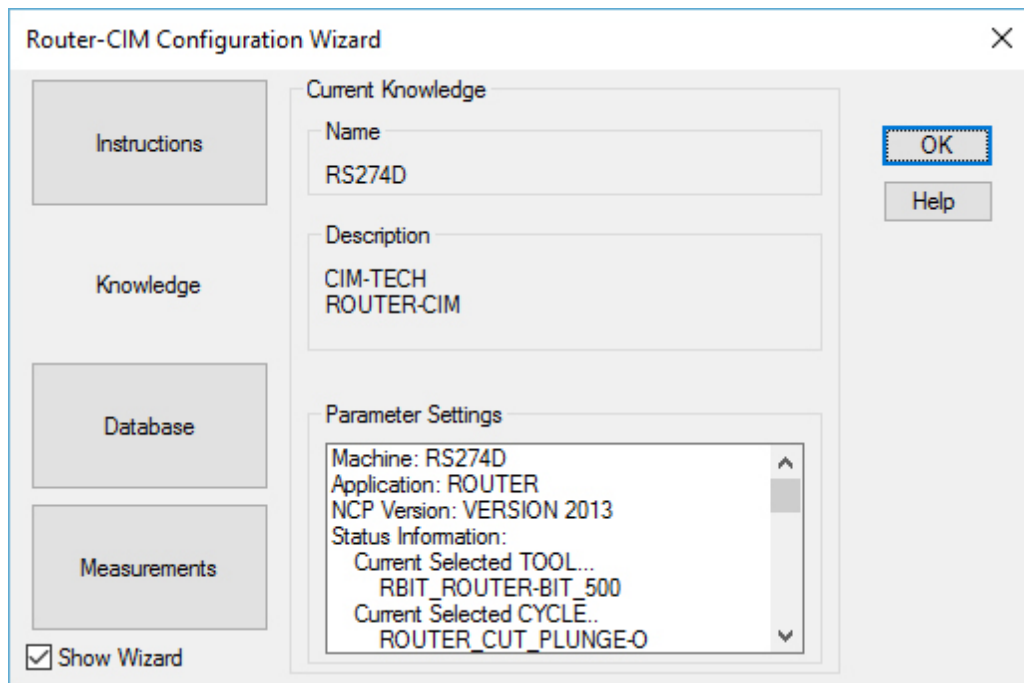


The Configuration Wizard

A knowledge base is created when a machine (Router) is linked to a postprocessor. A knowledge base name will always default to the name of the postprocessor. However, you are free to choose a name to your liking. Knowledge bases are saved as part of the current AutoCAD drawing.

As you use Router-CIM you will find that altering knowledge base settings and/or defaults to suit your particular situation can be advantageous. These alterations can be saved under a new knowledge base name. The Configuration Wizard is your tool to managing knowledge bases.

The Configuration Wizard dialog appears when Router-CIM is launched. The Current Knowledge base appears in the grayed out area at the center of the dialog screen. It contains the Name and Description of the knowledge base that was last defined by the Configuration Wizard, or it will display the most recently dated postprocessor found when Router-CIM is started for the first time.



View the Parameter Settings in the window of the dialog screen. If the Machine and/or Application parameters are not what you want for a configuration, use the Database button to retrieve a different machine or postprocessor.

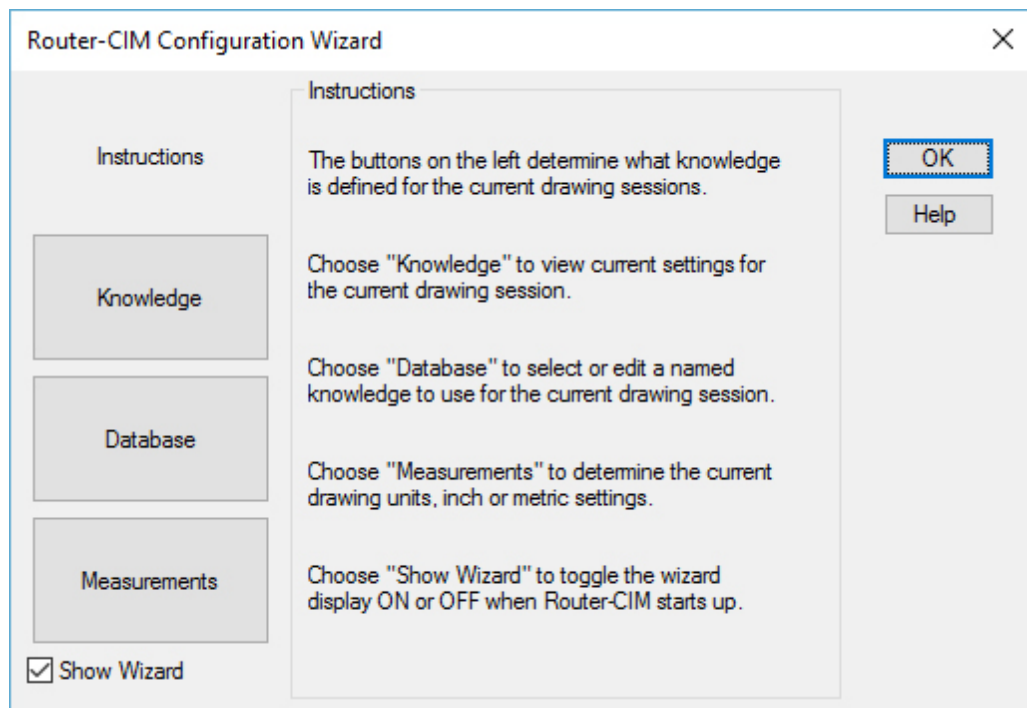
See ["How to Configure Router-CIM for Your Machine"](#) for step-by-step instructions.

To customize knowledge base settings and/or defaults see ["Customizing the Knowledge Base"](#).

Click **'OK'** to exit the Configuration Wizard and establish the selected knowledge base in the current drawing session. After clicking **'OK'** you can return to the Configuration Wizard to select a new postprocessor or to modify a knowledge base. To return to the Wizard, you must go to the RCIM pull-down menu and choose Configuration Wizard.

Instructions

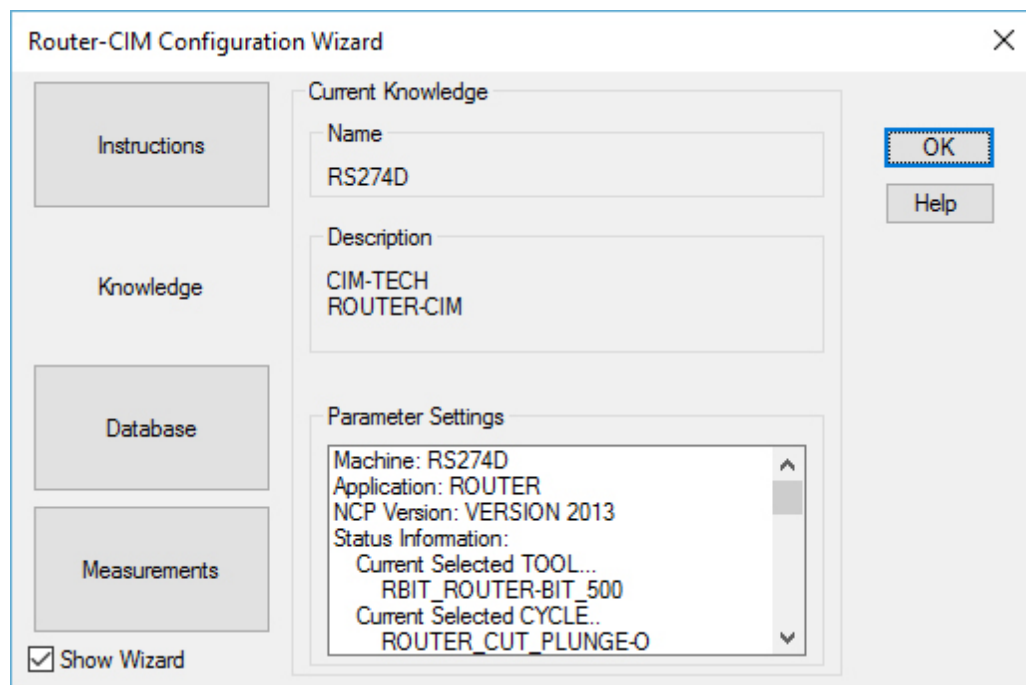
Choose **'Instructions'** to see a brief description of each button choice.



Knowledge

Knowledge is the default choice when the Configuration Wizard appears.

This window will show information for the currently selected post processor.

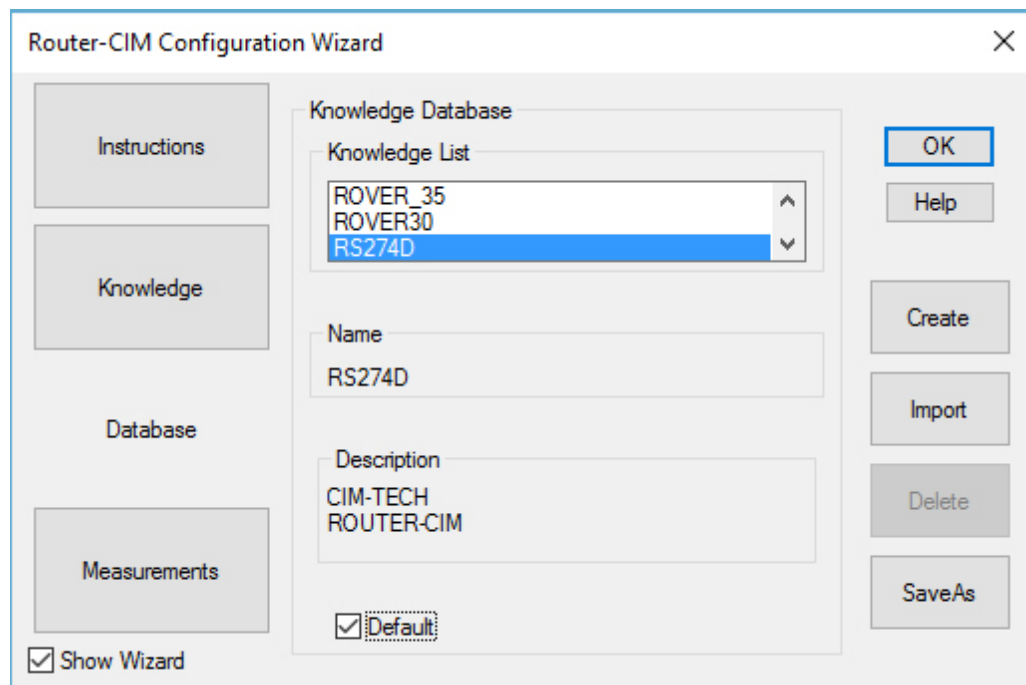


The Name and Description fields contain the information found in the postprocessor file as it was delivered to you. If you have made custom knowledge bases, and made one of them the Default, it will display the information for that system.

The Parameter Settings are obtained by the system, so information in this screen cannot be edited. Status Information displays the default configuration for the current knowledge base and it displays information such as the Machine Post name and the version of Router-CIM the post and post drawing were configured from. The default Tool, Cycle and Status information for the knowledge drawing are shown here as well.

Database

Click **'Database'** to display the available Knowledge List, Name and Description for any named knowledge bases you may have created or installed. In Database mode, new buttons appear that allow you to Create, Import, Delete and Save knowledge bases for use in Router-CIM.



The screenshot shows the 'Router-CIM Configuration Wizard' window with the 'Database' tab selected. On the left is a sidebar with buttons for 'Instructions', 'Knowledge', 'Database' (selected), and 'Measurements'. At the bottom of the sidebar is a checkbox labeled 'Show Wizard' which is checked. The main area is titled 'Knowledge Database' and contains a 'Knowledge List' dropdown menu with three items: 'ROVER_35', 'ROVER30', and 'RS274D' (which is highlighted in blue). Below the list are fields for 'Name' (containing 'RS274D') and 'Description' (containing 'CIM-TECH' and 'ROUTER-CIM'). At the bottom left of the main area is a checkbox labeled 'Default' which is checked. On the right side of the window are several buttons: 'OK' (highlighted with a blue border), 'Help', 'Create', 'Import', 'Delete', and 'SaveAs'.

The Database screen shows the available Knowledge database. Here you can change from one knowledge base to another. If you are just starting a session, and there are multiple entries in the Knowledge List, you can change from one machine to another.

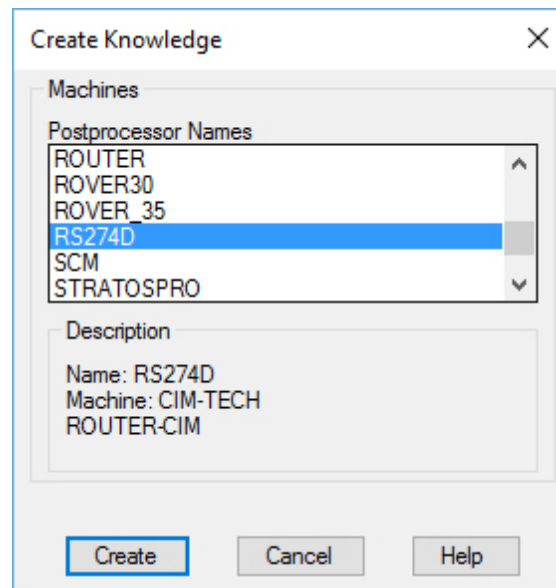
By selecting a machine from the Knowledge List, all available post processors and named knowledge bases for the selected machine appear. If no post processors have ever been accessed by the Configuration Wizard, you will be asked to create a named knowledge base by selecting a postprocessor. This will create a Knowledge List of one knowledge base using the selected postprocessor. If you have additional post processors you can create additional knowledge bases using the Create button. The Create button will generate additional knowledge bases and they will appear in the

Knowledge List. By selecting an item in the Knowledge List, the Name and Description edit boxes are filled in.

You can select the buttons from the picture above to move to those topics.

Create Knowledge Base

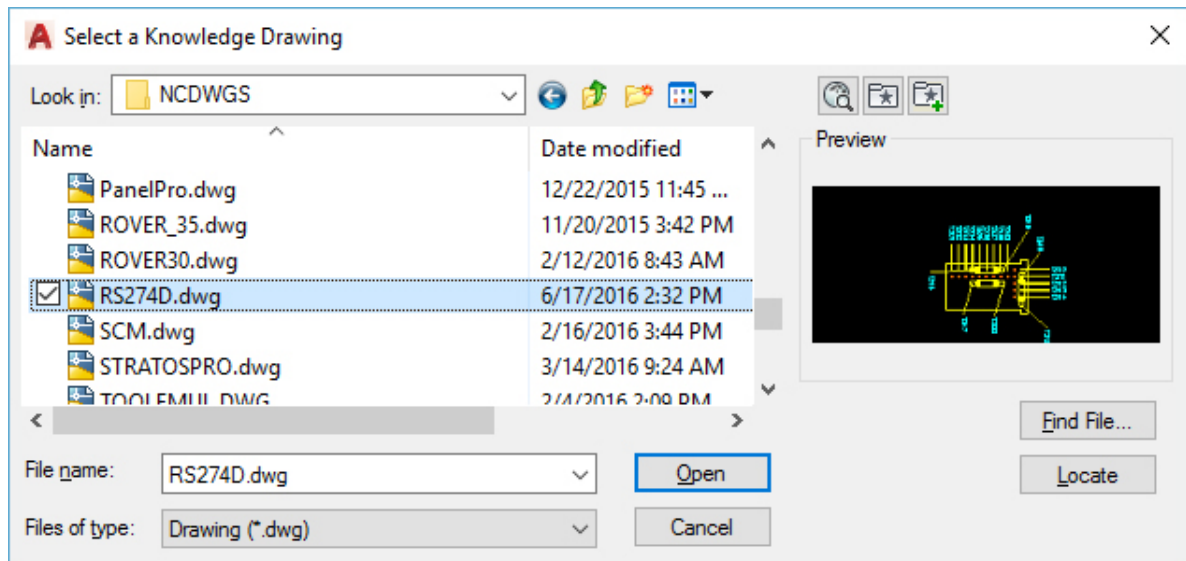
The **'Create'** button will open a selection window that allows you to create a new configuration from an existing configuration that is in the Router-CIM\Ncpost folder.



The **'Create'** button creates a new knowledge base, based on the information found in a selected postprocessor. Select the desired postprocessor name and the knowledge base is created. The Configuration Wizard will return you to the Database display.

Import Knowledge Base

Import will open a file selection window and you can select a knowledge drawing from there to define a new configuration. Any drawing that Router-CIM has been successfully run in and saved is potentially a new knowledge drawing.

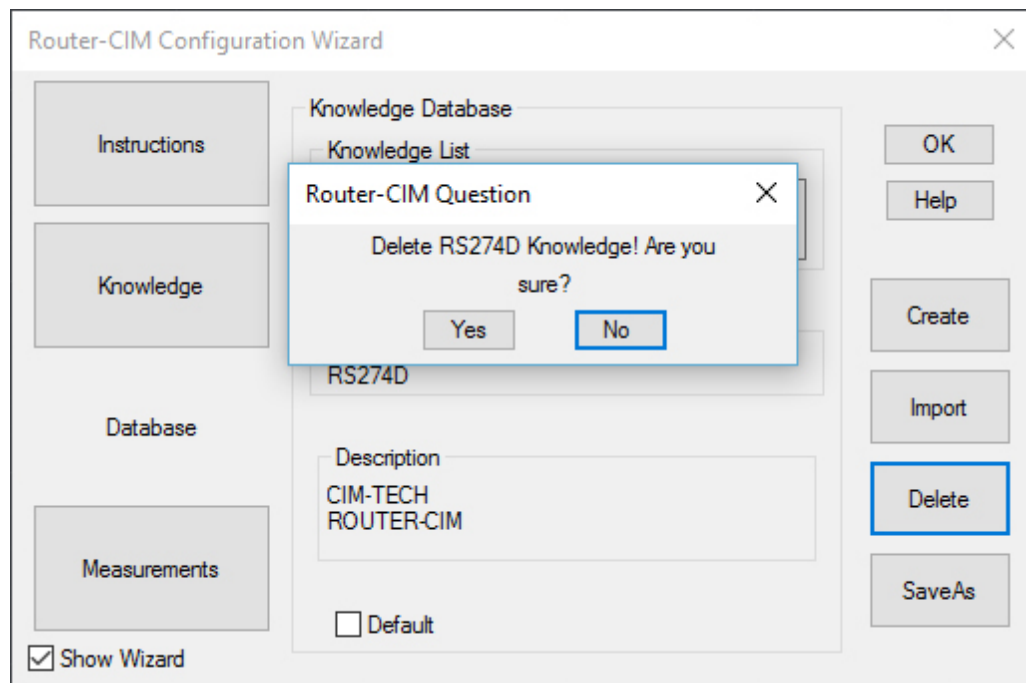


The **'Import'** button imports a knowledge base defined in another drawing. Existing Router-CIM users can import the knowledge bases used in previous versions of Router-CIM. Select a drawing that you know has a knowledge base, and the Knowledge list will be updated with the imported knowledge base using the name of the drawing selected. If the selected drawing has a knowledge base that does not agree with the current machine choice, the knowledge base in the selected drawing will be rejected.

Delete Knowledge Base

The Delete button is the opposite of the **'SaveAs'** button. The **'Delete'** button removes named Knowledge Bases from the configuration database.

This function will allow you to delete a selected knowledge base from the list. You will be prompted to be sure that this is the action you wish to perform.

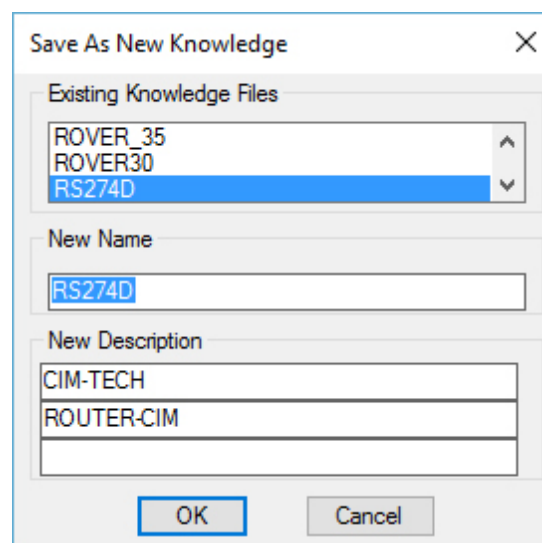


Once a knowledge base is deleted, you will have to create or import it and save it again to get it back.

If a Default knowledge is deleted, the Router becomes the default the next time Router-CIM loads.

Save Knowledge Base

The '**SaveAs**' button saves a selected knowledge base by name into the configuration database. The configuration database contains all the named knowledge bases that you see in the Knowledge List for each machine.



Existing Knowledge Files

The Existing Knowledge Files list displays the existing knowledge bases available for the current machine.

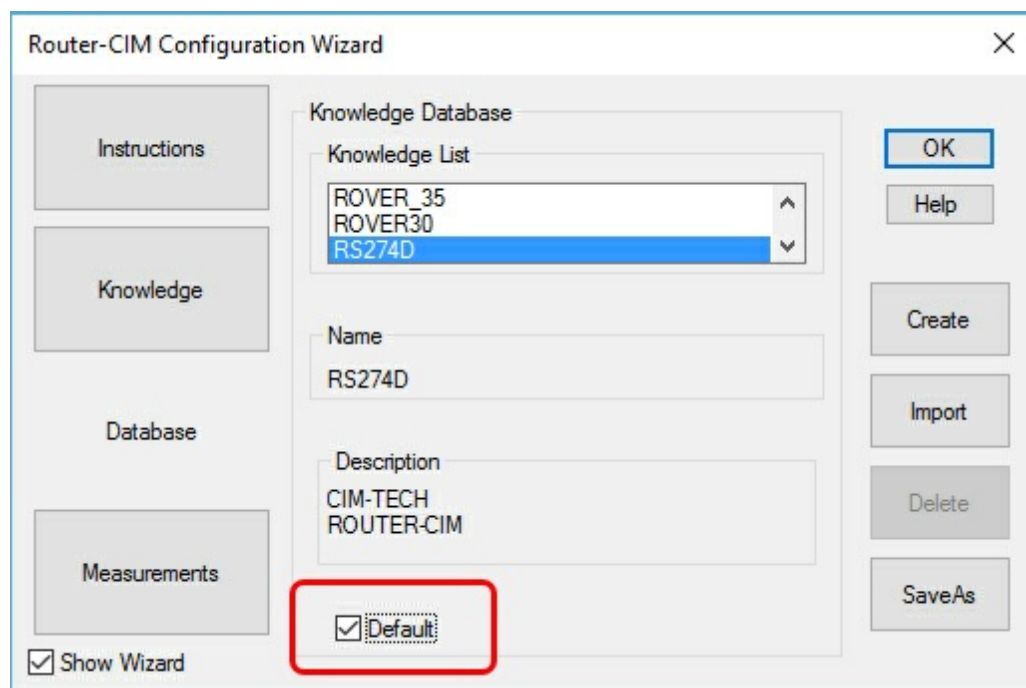
New Name

The New Name edit box is used to create a new knowledge base name for the Knowledge List. By typing a new name in the edit box and clicking 'OK', a new knowledge base name is created in the Knowledge List. This new name will have the same Description as the original unless you edit the Description fields.

New Description

The New Description fields are used to describe the named knowledge base. There are three lines of information available.

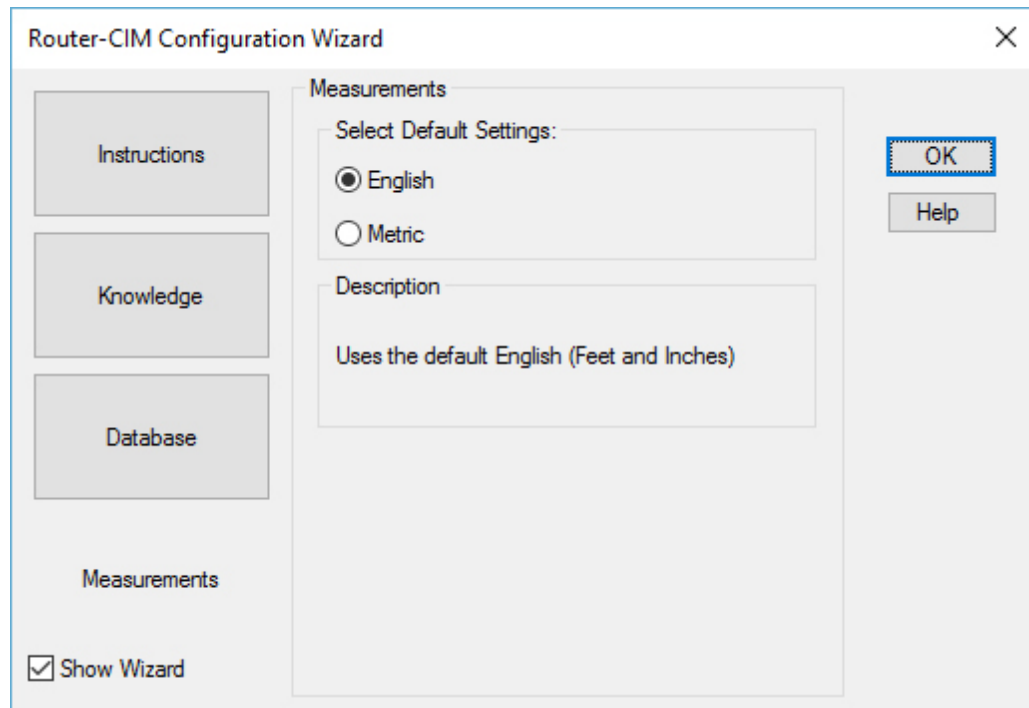
Default Knowledge Base



The Default check box indicates which knowledge base will be the default when Router-CIM starts. Check this box when the desired knowledge base is selected in the Knowledge List. Only one default can exist at any time. When you check this box, the previous default is removed and a new one established for the current named Knowledge.

Measurements

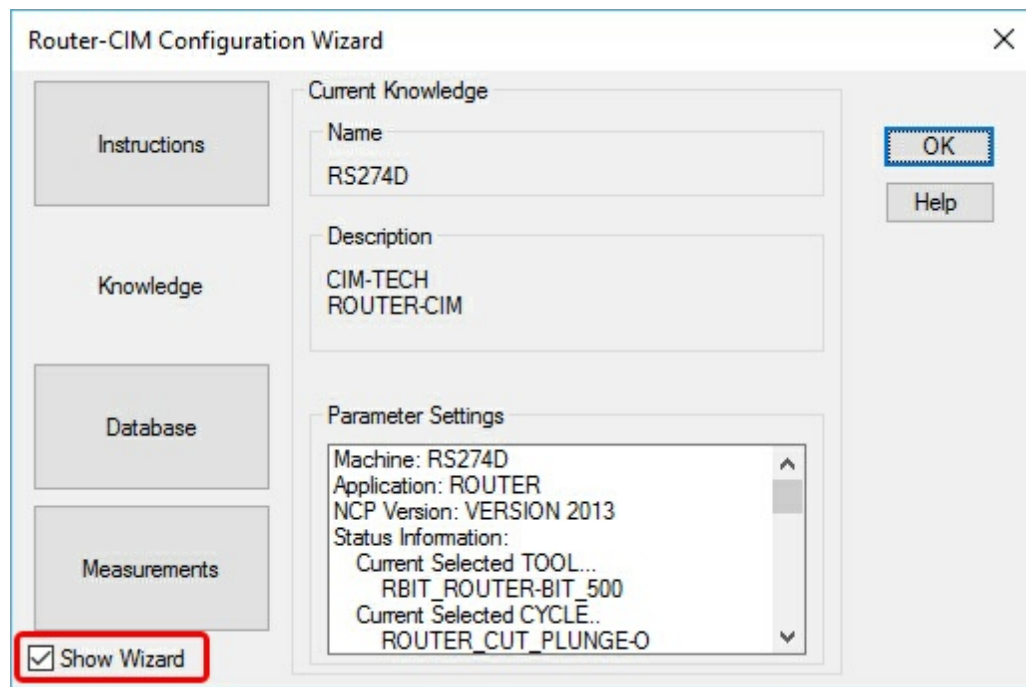
This button allows the setting of metric or inch units for the current drawing session. All the Router-CIM variables that depend on units are updated to this selection. If a drawing already exists, the units are predefined. You can change the unit settings, but they will not agree with the settings of the original drawing. The metric or inches setting in NCVARs is not affected. In other words, changing this setting will not scale your drawing, but only change the default units that the post processor reports when making NC Code.



Show Wizard

Choosing this will allow the Configuration Wizard to be displayed each time a Router-CIM session is begun. If this is unselected (remove check) then Router-CIM will load without the Configuration Wizard each time a drawing is begun that has a knowledge base in it.

The wizard will still be displayed when no knowledge is present in the drawing.



The 'Router-CIM Configuration Wizard' dialog box is shown. On the left, there are four buttons: 'Instructions', 'Knowledge', 'Database', and 'Measurements'. Below these is a checkbox labeled 'Show Wizard' which is checked and highlighted with a red rectangle. The main area on the right is titled 'Current Knowledge' and contains three sections: 'Name' with the value 'RS274D', 'Description' with the value 'CIM-TECH ROUTER-CIM', and 'Parameter Settings' which lists machine and application details. At the bottom right of the main area are 'OK' and 'Help' buttons. The 'OK' button is highlighted with a blue dashed border.

Router-CIM Configuration Wizard

Instructions

Knowledge

Database

Measurements

☒ Show Wizard

Current Knowledge

Name
RS274D

Description
CIM-TECH
ROUTER-CIM

Parameter Settings

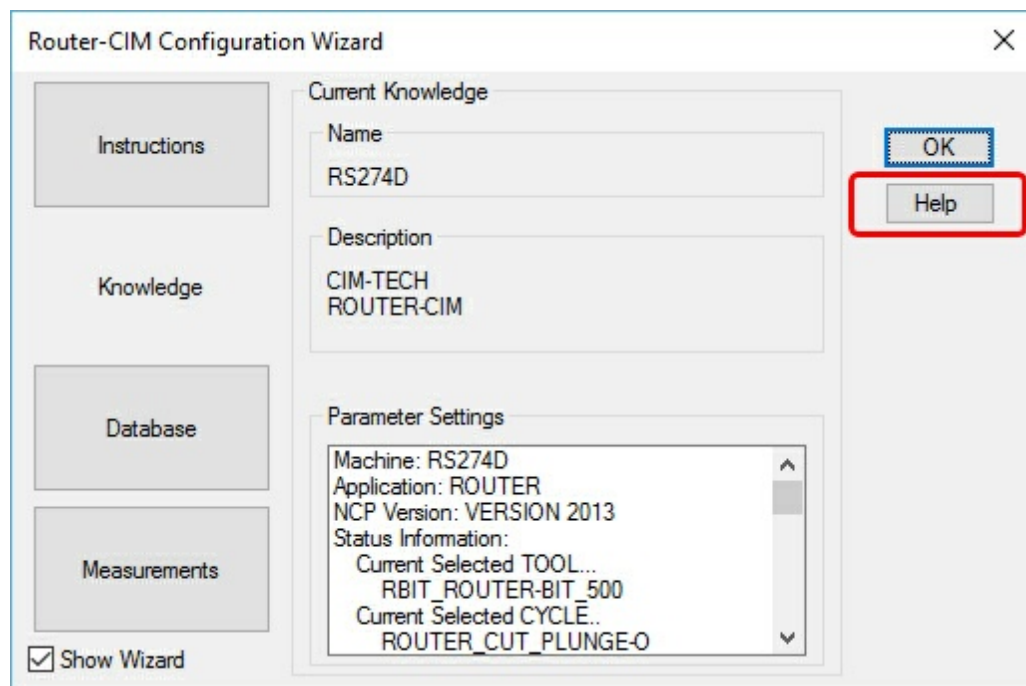
Machine: RS274D
Application: ROUTER
NCP Version: VERSION 2013
Status Information:
Current Selected TOOL...
RBIT_ROUTER-BIT_500
Current Selected CYCLE..
ROUTER_CUT_PLUNGE-O

OK

Help

Help

Selecting the '**Help**' button displays a popup help covering the basic topics in this chapter.



This image shows the same 'Router-CIM Configuration Wizard' dialog box as the previous one. In this version, the 'Show Wizard' checkbox is still checked, but the 'Help' button at the bottom right is highlighted with a red rectangle. The 'OK' button remains highlighted with a blue dashed border.

Router-CIM Configuration Wizard

Instructions

Knowledge

Database

Measurements

☒ Show Wizard

Current Knowledge

Name
RS274D

Description
CIM-TECH
ROUTER-CIM

Parameter Settings

Machine: RS274D
Application: ROUTER
NCP Version: VERSION 2013
Status Information:
Current Selected TOOL...
RBIT_ROUTER-BIT_500
Current Selected CYCLE..
ROUTER_CUT_PLUNGE-O

OK

Help

How to configure Router-CIM for your machine

Included here are basic instructions on how to configure Router-CIM for your machine tool. It assumes that you have a post processor for your machine installed, and available to Router-CIM.

1. View the Database

After you launch Router-CIM and the Configuration Wizard dialog appears, click on **'Database'** to view the knowledge database. If the desired Knowledge is displayed, you are ready to begin using Router-CIM. If this is not the case, proceed to step 2.

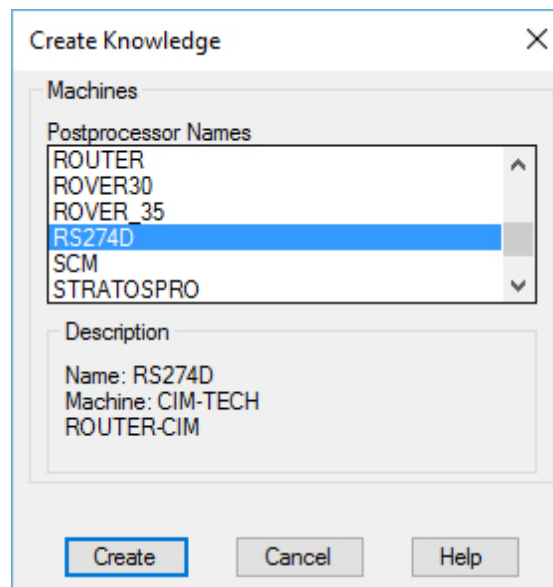
2. Select a Machine

If the correct machine is displayed (Knowledge List) but the wrong postprocessor (Name) is displayed, proceed to step 3.

If the wrong machine is displayed (Knowledge List) select the correct machine from the list.

3. Create a New Knowledge Entry

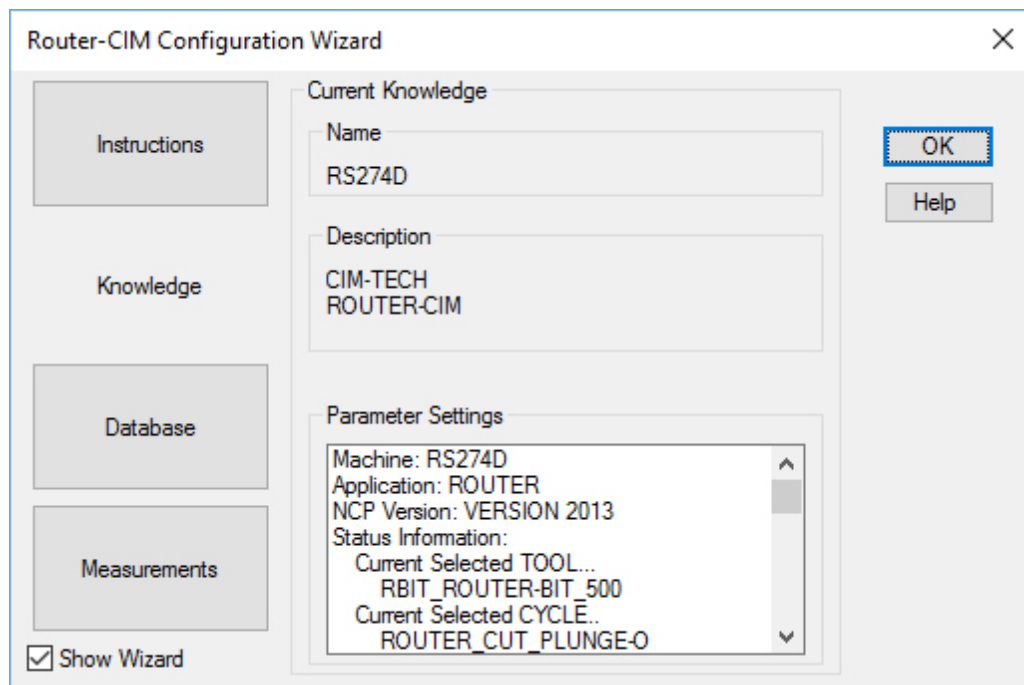
Click on **'Create'** to be presented with a list of all of the post processors that exist in your system.



Select the post processor from the list that fits your machine and click **'Create'** to link the selected post processor to the selected machine.

4. View the Information for the New Selected Knowledge.

After validating that the correct machine and post processor have been selected, click **'OK'** to use Router-CIM.

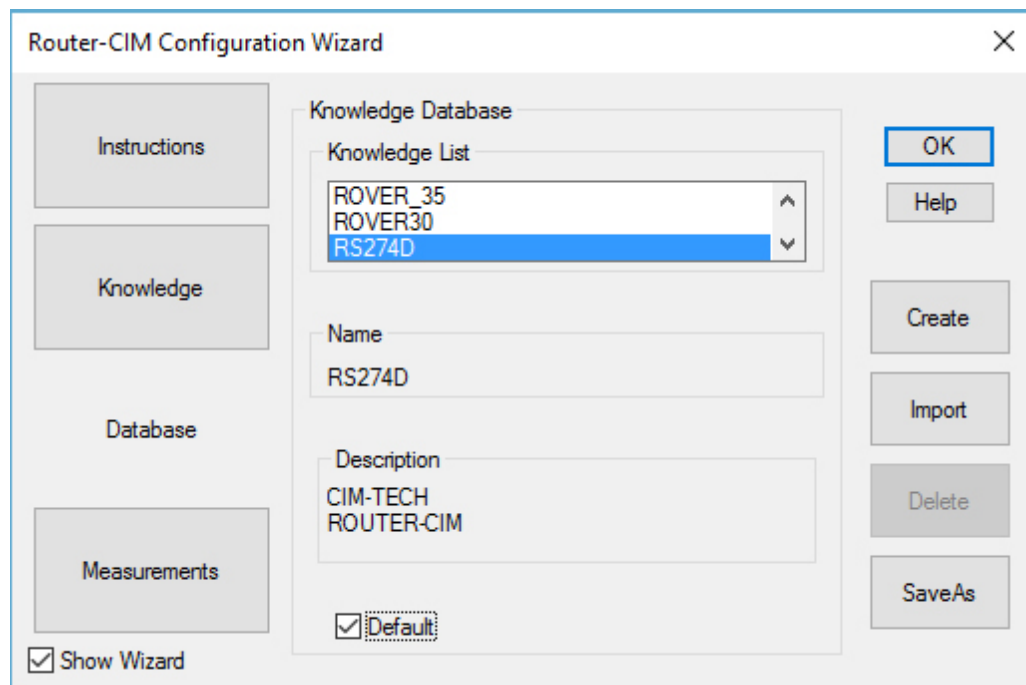


Customizing the Knowledge Base

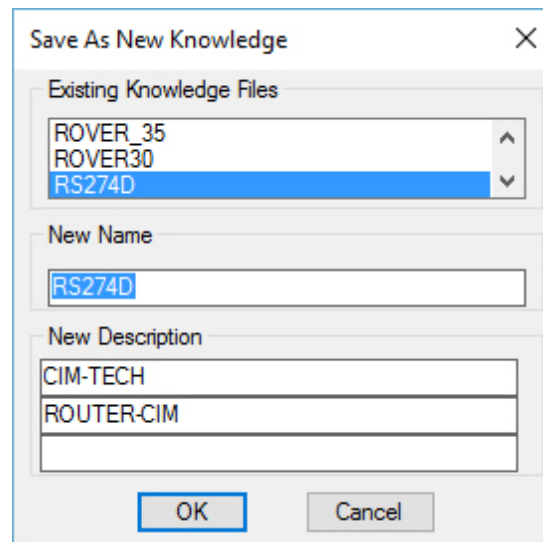
As you use Router-CIM you may find it desirable to change the system defaults of your knowledge base. To do so, make the desired changes to settings found in the Knowledge Editor, Tool information, Cycle information, or Status information. Use the Update Current Conditions command under NC Commands on the Router-CIM menu after each Cycle or Tool edit. This makes these changes available to be permanently saved by the Configuration Wizard.

When you have completed all of your edits, click on the Router-CIM pull-down menu and select **'Configuration Wizard'** to view your defaults.

If you want to save these settings to a named knowledge base, use the **'Database'** button.

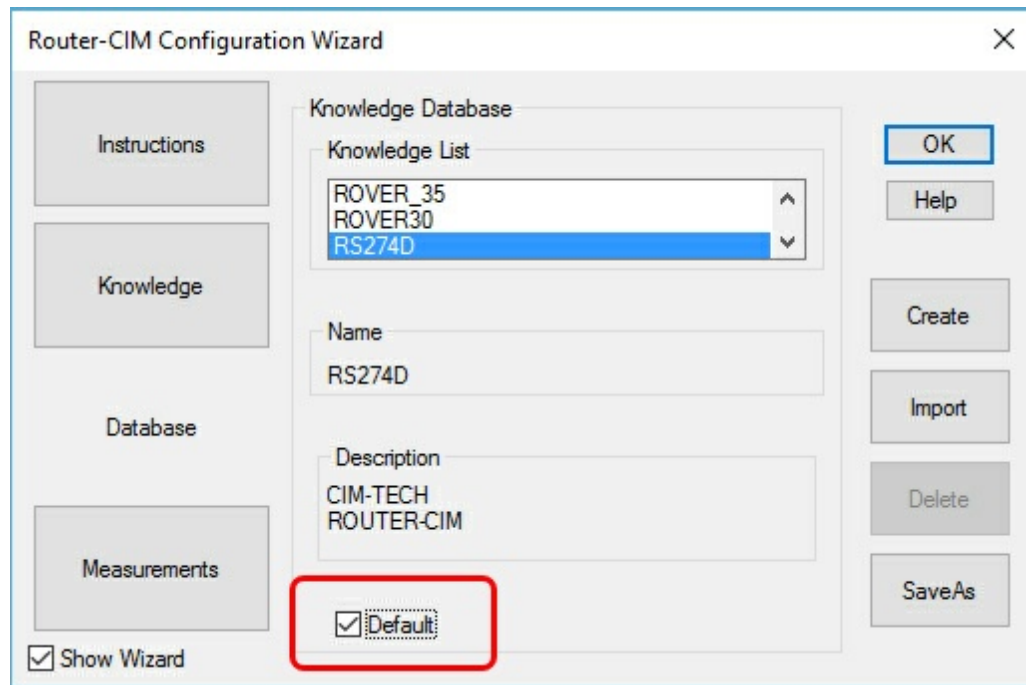


Click on '**SaveAs**'. When the dialog window appears, type a new Name and a new Description as appropriate.



Select '**OK**' when you are finished editing the Name and New Description.

If you would like this to become the default every time you launch Router-CIM, click the Default check box.



Router-CIM Control Panel

The Router-CIM Control Panel is where you define a cutting knowledge. A cutting knowledge is needed in order to apply a cut path. A typical knowledge will define the tool, the cutting cycle and the status information. The following sections will guide you on how to create, use and save a cutting knowledge.

CIM-Tech Router-CIM Control Panel

Description

Tool Information	Cycle Information	Status Information	Knowledge / Settings
ROUTER-BIT_500 Tools Tool Number / Comment "1" "ROUTER-BIT .5 DIA" CRC Offset: autocr Spindle Direction: CW Tool Diameter: 0.50000 Tool Radius: 0.25000 Tool Length: 4.00000 4 Axis Safe: Tool Type: Category: Vertical Offset: Horizontal Offset: Cutter Bridge: Aggregate Offset: <input type="radio"/> Spindle <input checked="" type="radio"/> Collet Cutter Compensation: <input type="radio"/> Yes <input type="radio"/> No <input checked="" type="radio"/> Both	Offset Dim: firstxyxcutloc Cut Side: outside Cut Direction: CW Round Corners: n Lead In: MULTILI Lead Out: MULTILO Lead Size: leadscl Lead Feed: Overlap Amount: AUTO XY Stock Allowance: Z Stock Allowance: Corner Punch: N	Safety Plane: 0.25000 Depth Per Pass: 1.00000 Total Cut Depth: Feederate/Spindle Speed Feederate: 1000.00000 Spindle Speed: 18000.00000 Surface FPM: NONE Units Per Revolution: NONE Ramp Feederate: Calculate Before Codes: After Codes: Oscillation Amount: 0.00000 Sort By Rank #:	Knowledge Select Knowledge CURRENT DOIT Edit Edit Retrieve Save Import Export Tabbing <input checked="" type="radio"/> No <input type="radio"/> Yes <input type="radio"/> Auto <input type="radio"/> Tab At Start <input type="radio"/> Tab By Distance <input checked="" type="checkbox"/> Acc-n-Dec <input type="checkbox"/> Metric <input type="checkbox"/> Inline <input type="checkbox"/> Plane Detect DOIT File: doitinfo.dat Tabbing Parameters Quantity: NONE Length: NONE Height: NONE Distance: NONE Minimum Radius: 0.00000 <input checked="" type="checkbox"/> Use 3D Tabs 3D Tab Ramp Amount: AUTO

PLUNGE OUTSIDE

Reset Cycle Settings to Default

Geoshape Group Start Point Cut Sequence Seq. Undo Edit Code Mod Cycle Mod Tool NCVars Last CutSIM CutSIM OK Close

Each heading in the Router-CIM Control Panel links to either the corresponding topic or the cutting cycle definition in the Router-CIM help manual.

Simply double-click on the heading next to the desired field and the Router-CIM help manual will open to that topic.

CIM-Tech Router-CIM Control Panel

Description

Tool Information

ROUTER-BIT_500

Tools

Tool Number / Comment

"1"

"ROUTER-BIT .5 DIA."

CRC Offset

Spindle Direction

Tool Diameter

Tool Radius

Tool Length

Geoshape, Group, and Start Point Edit

Geoshape

The Geoshape command is used to convert your drawing elements into usable 2D Polylines that Router-CIM uses to create tool paths. When you pick the Geoshape command, you will be prompted to select objects. Select items that represent the geometry that you wish to use to create a tool path or several tool paths from. Do not select objects that are not related to the tool path. You can use a window or crossing window, or a single pick selection. If using DO-IT, Pattern Recognition, Cutting a Solid, or a Surface; do not use Geoshape. The DO-IT command automatically defines shapes before Cutting.

The Geoshape command runs a task called Geoshape. The result of the Geoshape command depends on the geometry selected. The primary operation of this command is to make closed, counter-clockwise Polylines on layer NC_SHAPE from the selected geometry. Geoshape is the Router-CIM command that will changes lines, arcs and other geometry in Polylines. This is also the command that Router-CIM uses to blend or heal open or crossing geometry.

Group

Group is a command that lets you take several Geoshaped elements and place them into a single set or group. This way one cut command can be used on several shapes that are cut in the same way. Such as drilling many holes that are the same size. Grouping will work on holes, and on sperate pieces of geometry. There is no limit to the number of items in a group, but note that once the items are in a group, all of them are selected at once for any command given, like CUT.

Start Point Edit

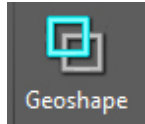
Start Point Edit is a command used to move or change the location where the tool starts it cut in a shape. Some start points are selected by default when the Geoshape command is run. These locations can be changed to suit your particular cutting needs.

Geoshape

Router-CIM Toolbar:



Router-CIM Ribbon:



Keyboard: GS

Router-CIM must have geometry to make toolpaths from. Valid types of geometry are Lines, Arcs, Circles, Polylines, 3D Polylines, Splines, Ellipses, and Solids.

Blocks should be broken down into usable geometry. There are many instances where the Geoshape command will do this for you, but nested blocks can cause the Geoshape command to leave some geometry in a block form and that is not acceptable for cutting.

Geoshape will also utilize text features that use an SHX font style. This will allow the user to keep the text as a font. If Geoshaped, the SHX text will be converted to lines and arcs for machining.

The first operation that the Geoshape command executes is to erase any existing objects on layer NC_GEO. This step is important because the objects on layer NC_GEO are used to make polylines on layer NC_SHAPE. These existing objects (if any) on NC_GEO are erased so that you don't get two or more NC_GEO objects on top of each other.

After the NC_GEO layer is cleared, Router-CIM issues the Geometry command. Geometry copies what you have selected onto layer NC_GEO. If you select Polylines or blocks, they will first be copied onto layer NC_GEO then exploded into their smallest element.

The second part of the Geoshape command is running the Shapes command.

The Shapes command now converts the NC_GEO objects into CCW Polylines on layer NC_SHAPE. The Polylines will be closed, if possible. The Geoshape command will fix contingencies or gaps within .02". This fix gap tolerance is adjustable in NCVARS, Geometric, *brng*.

These steps are necessary because only Polylines and surfaces on layer NC_SHAPE are suitable for cutting. Since there is blending, any lines that are not tangent or are open by more than .02" will be considered separate, open elements, and will generate their own start points (see the section on Start Point Edit later in this chapter for more information about start points). Gaps are fixed because the NCVAR called NCJOIN is set to True, and the NCVAR *brng* is set at 0.02 this is the largest gap to fix.

After completion of Start Point Edit, the Geoshape command will delete any geometry on layer NC_GEO. The Geoshape command will place NC_SHAPE Polylines over the original objects. The original geometry produced is not altered in any way by Router-CIM.

Since the qualities of your tool paths are determined by your drawing, only clean, continuous objects should be selected. Proper geometry definition is important to making accurate tool paths. If you do not

like the shapes created, delete geometry on layer NC_SHAPE, fix the geometry on your original layer, and try again.

You can create your own Polylines on layer NC_SHAPE and use them for making tool paths as well. Certain rules apply if you make your own Polylines:

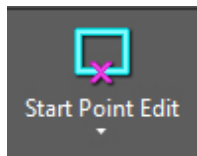
- Any polyline that you want to machine must be on layer NC_shape.
- If it is a closed shape it must go in the CCW direction.

Therefore, if you create a 2D polyline that goes in the CCW direction, you may place that polyline on layer NC_SHAPE (using either the AutoCAD Change or Chprop commands) and then Cut it.

Start Point Edit (Start Pt) and Start Point Control

Start Point Edit:

Router-CIM Toolbar: 



Router-CIM Ribbon:

Keyboard: SP

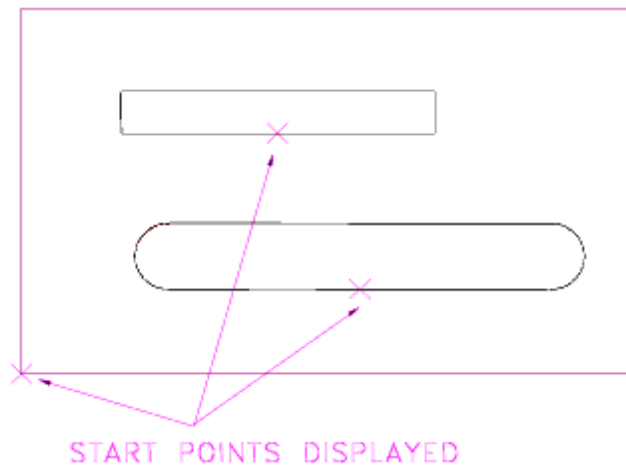
This command allows you to set or change the locations for the STARTING POINT of the cutting cycle. This is the point where the tool leads-in or leads-out of the material.

You will be prompted to select the items you wish to change the start points on, at this point the start points appear as magenta 'X's on the part.

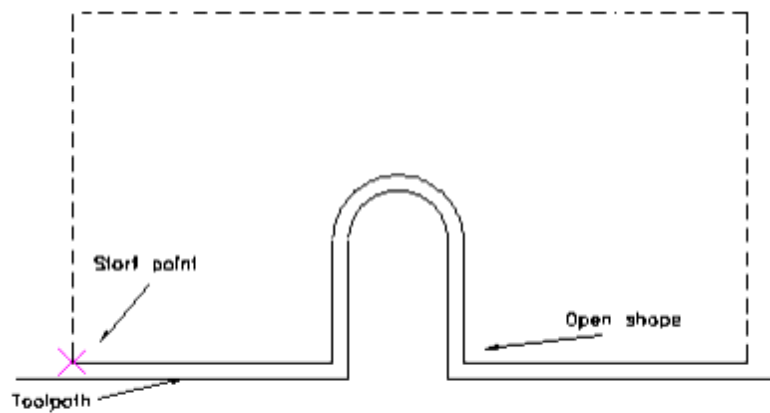
Multiple selection is allowed, so if there are several shapes you wish to change the start points on, select them all, and Router-CIM will highlight them one by one for you to select the start point positions.

If there are no start points you wish to change, then press <Enter> two times with no selection and you will exit the Start Point Edit command.

If you make a mistake picking a start point you can re-enter the Start Point Edit command by picking the StartPT button from the Control Panel, and selecting the geometry you wish to change start points on.

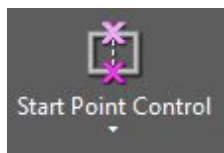


Start Point Edit will reverse the direction of an open shape (see figure 2). You can change the cycle's Cut Side and/or Cut Direction in the Control Panel to make a correct Cut on an open shape.



Start Point Control (PREDEF)

Router-CIM Toolbar: 

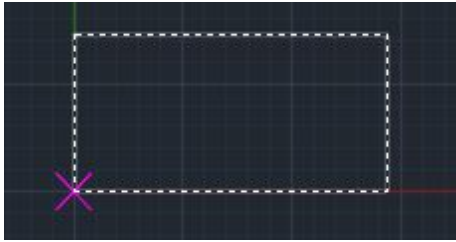


Router-CIM Ribbon:

Keyboard: **PREDEF**

This command allows you to set or change the locations for the STARTING POINT of the cutting cycle PRIOR to the GeoShape command. This point set during this command will become the default start point highlighted by the GeoShape Command.

You will be prompted to select the polyline you wish to define the start point location. At this point the start points appear as magenta 'X's on the part.



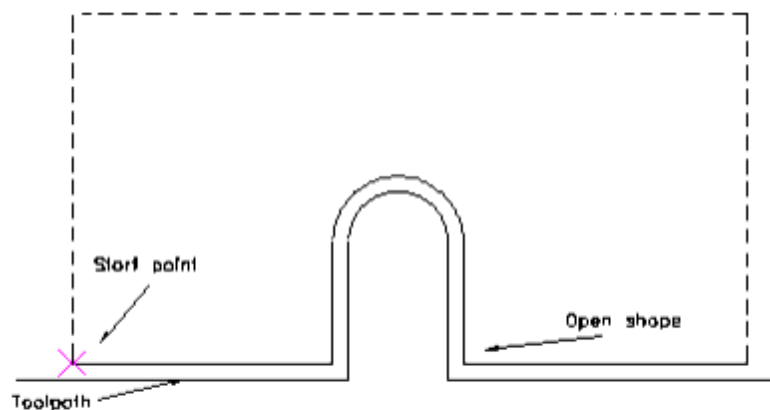
Multiple selection is allowed, so if there are several shapes you wish to define the start points on, select them all, and Router-CIM will highlight them one by one for you to select the defined start point positions.

Once you have defined the start points, then press <Enter> two times with no selection and you will exit the Start Point Control command.

If you make a mistake defining a start point you can re-enter the Start Point Control command by selecting the icons again or typing in PREDEF, and selecting the geometry you wish to define start points on.

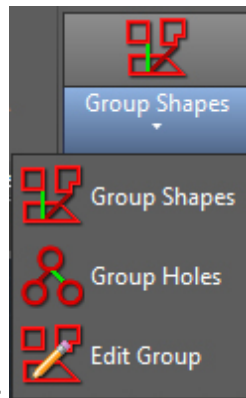
Open Shape Issues

The location of the start point indicates the break in the open shape. Drawings that have lines on top of other lines will create open shapes. If you were expecting a closed shape, but Start Point Edit does not let you change the start point, you have an open shape. Every time you pick your shape with Start Point Edit, it shows you the problem areas. The X indicates where the shape is not continuous, or has a gap that is larger than the gap tolerance (Geometric NCVAR brng). You can erase the open shapes (on layer NC_shape); fix the original geometry on the original layer, then Geoshape the objects again.



RCIM Group Command

Router-CIM Toolbar: 



Router-CIM Ribbon:

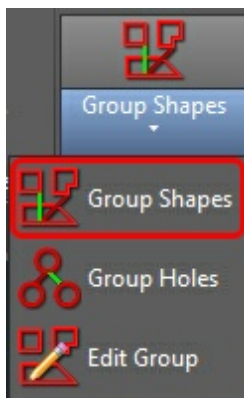
Above is the Group toolbar used for shapes or holes which have been previously Geoshaped. The icons, from left to right are: Group Shapes, Group Holes and Edit Group. The end result will block the shapes or holes together so only one item needs to be selected in the future. This is particularly helpful with large groups of holes or any combination of shapes you wish to cut using the same knowledge.

Group Shapes

The first icon in the Group toolbar is for Grouping Shapes. Following are the steps involved to group different shapes together so they may be cut with the same knowledge.

Note: If you group features together that will require multiple passes, the Group Shapes option will go to the first depth on all the shapes and then continue to the next depth on all the shapes until the shapes have been cut to the correct depth. Individual shapes will NOT be cut with all the passes needed prior to moving to the next shape.

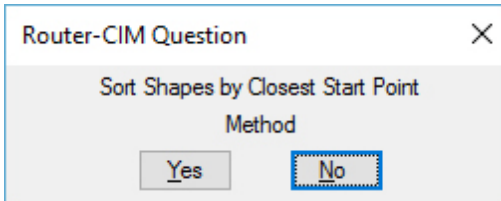
1. Geoshape the parts or pieces of geometry first.
2. Click on the '**Group Shapes**' icon.



At the command line you will be prompted:
SELECT Shapes to Group Together

Select Objects

3. Window around all the object you want in a group or pick them in the order you wish them to be cut. Left Click then Enter. You will be prompted with:

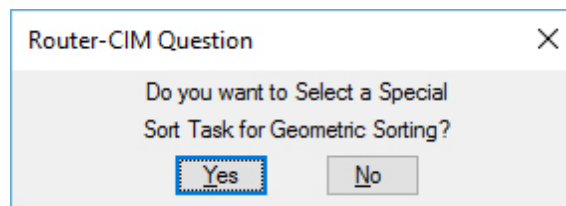


The shapes will turn **Red**. Choose **'Yes'**. You may choose **'No'** if you have only a few shapes which you have already chosen in the order you want them cut.

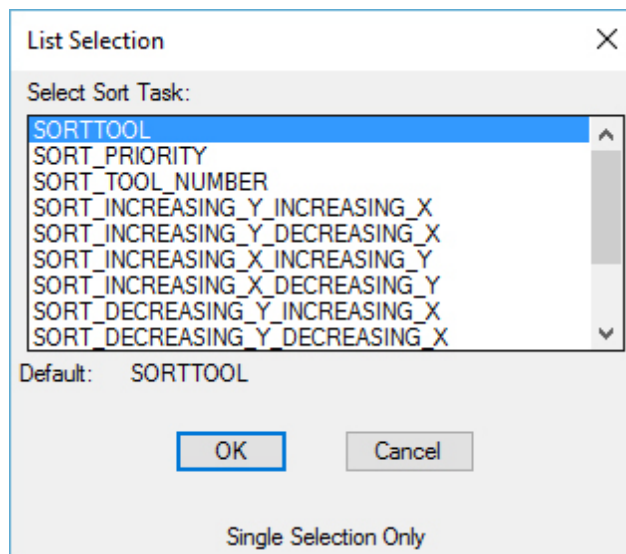
4. At the command line:

Choose First Shape in Packet Select Objects

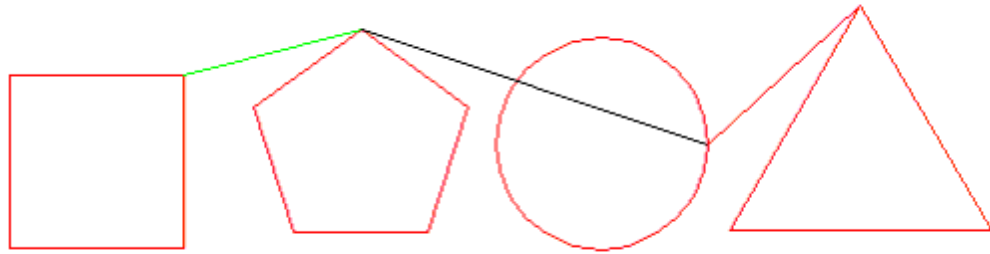
Pick on which one you want to be first in the group. Then Enter. You will be prompted with:



Choosing **'No'** will sort the shapes by Closest Start Point method. Choosing **'Yes'** will bring up a dialog box allowing you to choose different sorting methods:



5. After selecting either **'No'** or Choosing **'Yes'** followed by a different sorting method, your parts will be grouped together enabling you to cut them with the same knowledge.

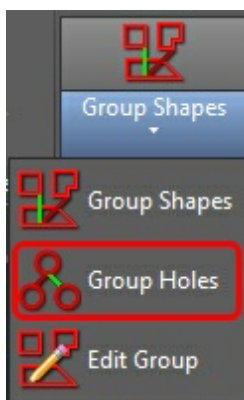


In the picture above the **Green** line coming off the square indicates the square is the first object in the group and the **Red** line leading to the triangle indicates the triangle is the last object in the group.

Group Holes

The second icon in the Group toolbar is for Grouping Holes. This option can be very helpful in situations with line boring or any operation involving a large amount of holes.

1. Geoshape the holes first.
2. Click on the Group Holes icon

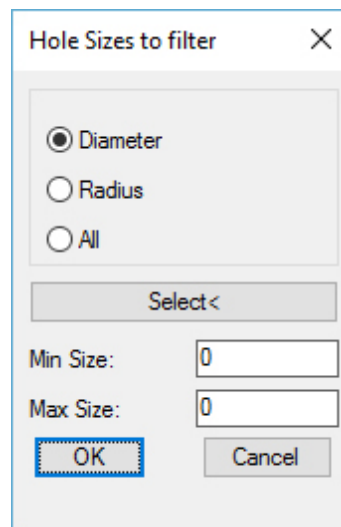


At the command line you will be prompted:

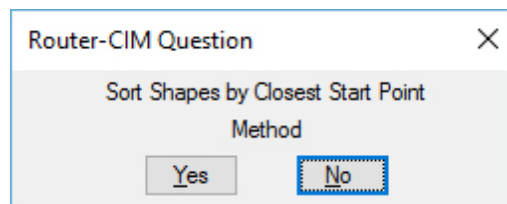
Select shapes to search for circles:

Select objects:

3. Window around all the holes in your group or pick them in the order you wish them to be cut. Left Click, then Enter. You will be prompted with the following dialog:



4. At this point you may select the holes in two different ways. If you know all of your holes are the same size and you want them in one group you may change the hole parameters from "Diameter" or "Radius" to "All" and then click on '**OK**' and you will be prompted with:

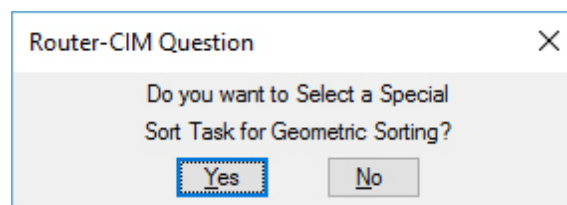


5. The holes will turn **Red**. Choose '**Yes**'. You may choose '**No**' if you have only a few holes which you have already chosen in the order you want them cut.

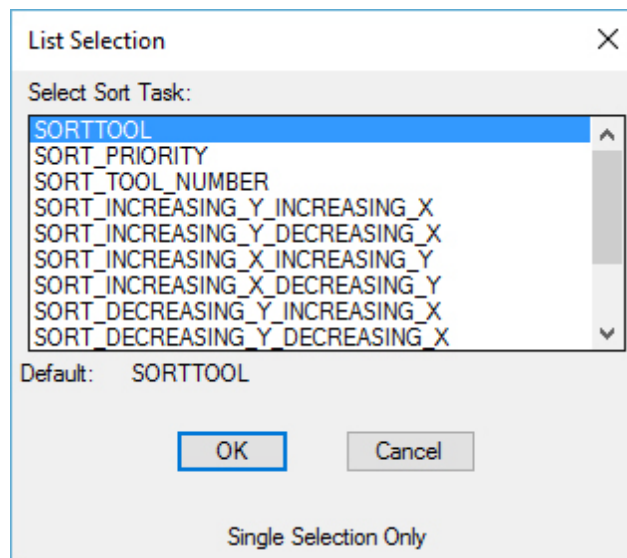
6. At the command line:

Choose First Shape in Packet
Select Objects

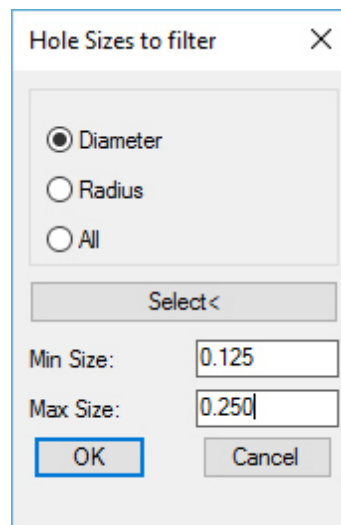
Pick on which one you want to be first in the group. Then Enter. You will be prompted with:



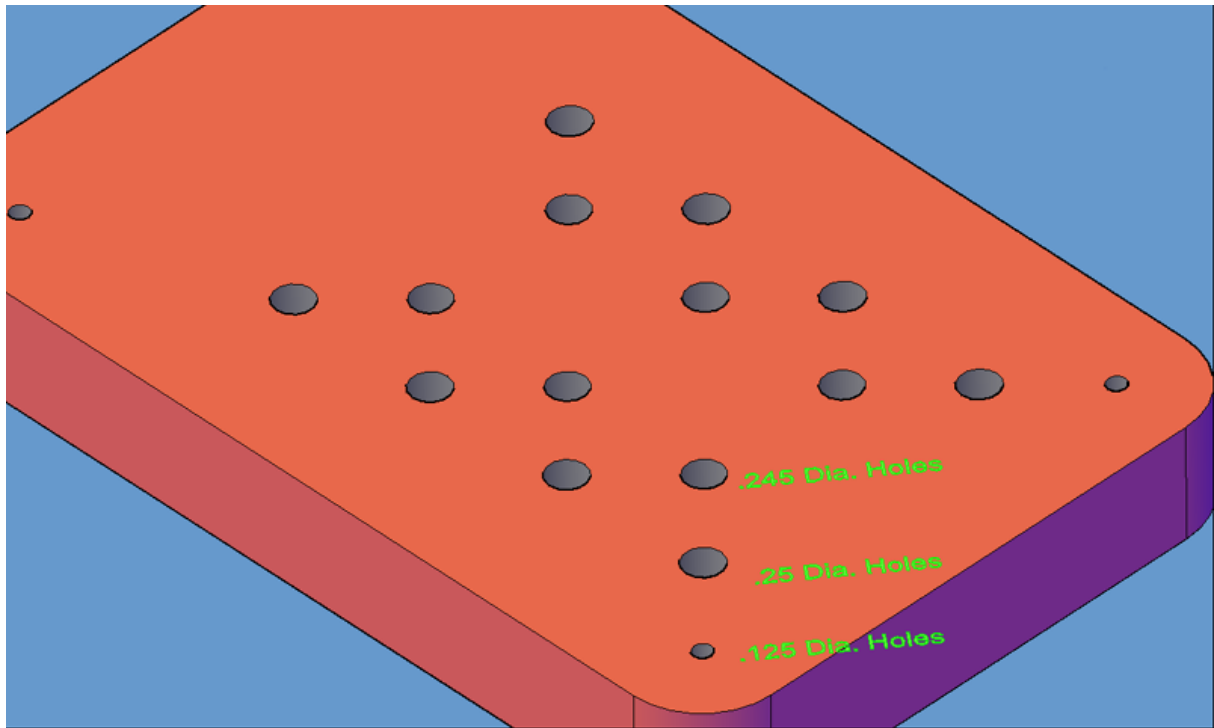
Choosing '**No**' will sort the shapes by Closest Start Point method. Choosing '**Yes**' will bring up a dialog box allowing you to choose different sorting methods:



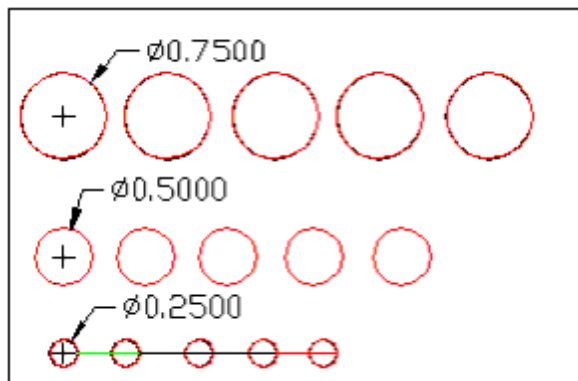
7. If you have a group of holes which are close to the same size, but differ by a small amount and you wish to have them all grouped together, you may do this by using the "Min Size" "Max Size" setting to search for holes within a specified range.



If you had a few holes which happened to be 0.125 in diameter in addition to the ones which are 0.245 and 0.250 and you wanted to center drill all of them at once, you could enter these values into the "Min Size" and "Max Size" fields and Router-CIM will search for them and include them in the group.



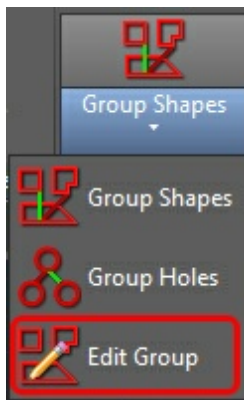
Note: If you have multiple groups of holes in a part which differ in size (i.e. .25, .50, .75 etc.) you can create each group individually based on size if that is preferred.



Edit Group

This is the third icon on the Group Shapes tool bar and allows you to Edit the features of a previously created group. This function allows you to Add, Remove, or Sort objects of an existing group.

1. After you have determined which group it is you want to edit. Click on the **'Edit Group'** icon.



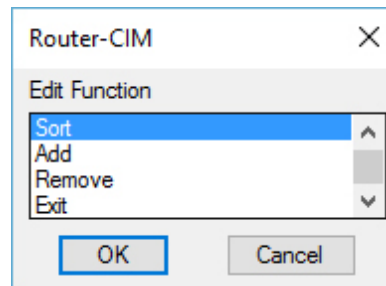
You will be prompted with:

Select Shape Pack to Edit:

2. Pick on one of the objects which is part of the group.

If you try to pick on the line connecting the pieces, the group will not be recognized.

3. After you select one you will be prompted with the following dialog box:



Draw

The Draw function will re-draw the group sorting line on the selected group. The sorting line shows which shape is first (**green**) which shapes are next (**white**) and which shape is last (**red**).

Sort

Allows you to pick a different sorting method for the group. By default, the objects will be sorted by the Closest Start Point method.

Add

If you have added additional objects to the drawing you are working on and wish to add them to a current Group, you may do so with this feature. Click on Add, then OK and you will be prompted with

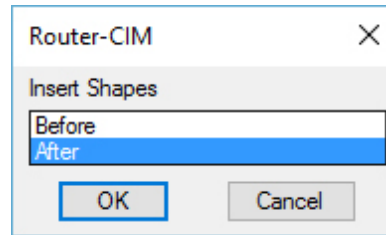
Select Shapes to ADD

Select Objects

Choose the new shape(s) and Enter.

Select Position Shape

The Position shape is the object in the current Group you want to use as a reference for adding the new one. Once selected you will be prompted with a new dialog:



Choose the positioning you want and the object will be added to the existing group.

Remove

If you wish to remove objects from a Group in the current drawing you are working on, you may do so with this feature. Click on Remove, then OK and you will be prompted with

Select Shapes to Remove **Select Objects**

Click on the one(s) you wish to Remove, then hit Enter. The shape will turn from Red back to Blue (Geoshape color) and will no longer be part of the Group.

Troubleshooting Geometry

Some geometry could cause issues that are not readily detectable until the part is run on the machine. Other geometry can cause issues that are readily apparent when a tool path is created.

Crossing Geometry:

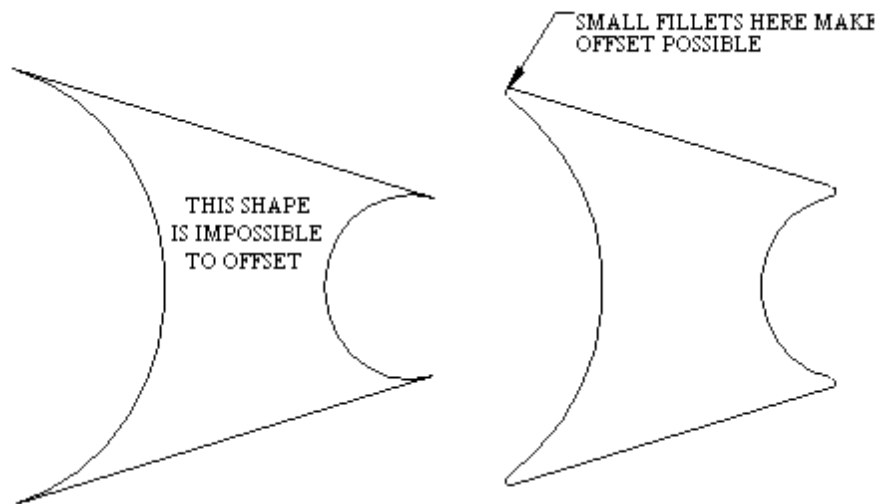
It is important to note that lines and arcs that cross other lines and arcs could confuse the Shapes command.

When lines and arcs have the same end points, the shape could go in several directions. Keep all geometry to be Geoshaped clean and continuous without ambiguous end points.

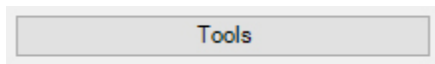
The easiest way to see incorrectly shaped geometry is with the Start Point Edit command. If you have a shape that is supposed to be closed and Router-CIM is finding more than one Start Point, or if the Start Point will not change, then the geometry is not closed and should be repaired before going any further. The location of the X is where a problem exists in the geometry.

Impossible Offsets:

Certain geometry is impossible to offset. Under certain circumstances, you may have to insert a Fillet or Chamfer to allow offset. These situations occur when arcs and lines cannot connect when offset. The Fillet or Chamfer will usually solve this problem. Use the smallest acceptable radius or distance.



Tool Information



Following is a brief explanation of Tool Information, the first column in the Control Panel/All Stats page.

Select a tool from the Rbit Tool List by picking on the Tools Button in the Control Panel.

Tool Information

ROUTER-BIT_500

Tools

Tool Number / Comment

"1"

Add Edit

"ROUTER-BIT .5 DIA."

CRC Offset autocr

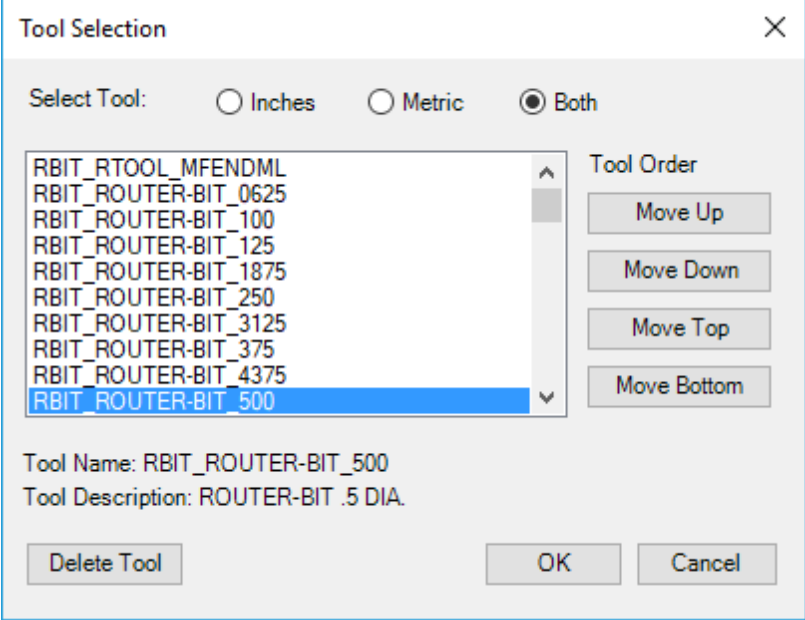
Spindle Direction CW

Tool Diameter 0.50000

Tool Radius 0.25000

Tool Length 4.00000

A dialog box showing the tool names found in the Rbit list will appear. Move the cursor over the tool to use and pick the tool you wish to Cut with.



The image shows a 'Tool Selection' dialog box. At the top, there is a 'Select Tool:' label followed by three radio buttons: 'Inches', 'Metric', and 'Both'. The 'Both' radio button is selected. Below this is a list box containing several tool names: 'RBIT_RTOOL_MFENDML', 'RBIT_ROUTER-BIT_0625', 'RBIT_ROUTER-BIT_100', 'RBIT_ROUTER-BIT_125', 'RBIT_ROUTER-BIT_1875', 'RBIT_ROUTER-BIT_250', 'RBIT_ROUTER-BIT_3125', 'RBIT_ROUTER-BIT_375', 'RBIT_ROUTER-BIT_4375', and 'RBIT_ROUTER-BIT_500'. The last item, 'RBIT_ROUTER-BIT_500', is highlighted in blue. To the right of the list box is a 'Tool Order' section with four buttons: 'Move Up', 'Move Down', 'Move Top', and 'Move Bottom'. Below the list box, the 'Tool Name' is displayed as 'RBIT_ROUTER-BIT_500' and the 'Tool Description' is 'ROUTER-BIT .5 DIA.'. At the bottom of the dialog are three buttons: 'Delete Tool', 'OK', and 'Cancel'.

Tool Selection

Select Tool: ☐ Inches ☐ Metric ☒ Both

RBIT_RTOOL_MFENDML
RBIT_ROUTER-BIT_0625
RBIT_ROUTER-BIT_100
RBIT_ROUTER-BIT_125
RBIT_ROUTER-BIT_1875
RBIT_ROUTER-BIT_250
RBIT_ROUTER-BIT_3125
RBIT_ROUTER-BIT_375
RBIT_ROUTER-BIT_4375
RBIT_ROUTER-BIT_500

Tool Order

Move Up
Move Down
Move Top
Move Bottom

Tool Name: RBIT_ROUTER-BIT_500
Tool Description: ROUTER-BIT .5 DIA.

Delete Tool OK Cancel

The system will use the current attributes of the tool you select.

The screenshot shows a 'Tool Information' dialog box. At the top, it displays 'ROUTER-BIT_500'. Below this is a 'Tools' button. A 'Tool Number / Comment' section contains a dropdown menu with '1' selected, and 'Add' and 'Edit' buttons. Below that is a text field containing 'ROUTER-BIT .5 DIA.'. The main section contains several input fields: 'CRC Offset' (autocrc), 'Spindle Direction' (CW), 'Tool Diameter' (0.50000), 'Tool Radius' (0.25000), 'Tool Length' (4.00000), '4 Axis Safe', 'Tool Type', 'Category', 'Vertical Offset', 'Horizontal Offset', and 'Cutter Bridge'. At the bottom, there are two sections: 'Aggregate Offset' with radio buttons for 'Spindle' and 'Collet' (selected), and 'Cutter Compensation' with radio buttons for 'Yes', 'No', and 'Both' (selected).

The standard Tool list contains most of the common size router bits, drill bits, ball mills, as well as some generic shaping tools (e.g. round over bits).

These tools are listed in English Units at the top half of the list and in Metric Units at the bottom half of the list.

You can switch between the tools shown by selecting the measurement of tools you want to see:

The screenshot shows a 'Select Tool' dialog box with three radio buttons: 'Inches', 'Metric', and 'Both'. The 'Both' button is selected.

After selecting a tool and selecting the 'OK' button, the Control Panel will reflect all the current information for that selected tool in the Tool Information column.

This list of tools is may be customized by the user, and can contain all the tools you will use on a regular basis. You may add tools to any position in the list, however setting or keeping a specific order to the list is usually helpful in finding and/or changing tools in the future.

Tools can be ordered in the list by highlighting the specific tool and using the **'Move Up'**, **'Move Down'**, **'Move Top'** or **'Move Bottom'** buttons on the right side of the dialog box.

To delete a tool from the list, highlight the specific tool and select the 'Delete Tool' button on the lower left of the dialog box.

Tool Number and Comment

The top field is the Tool Number field, and indicates any tool or spindle numbers that are to be used for the current cut.

The bottom field is the Tool Comment field, and indicates any comment you want placed in the code at the start of the cut. This field will also represent the tool in the [Summary Report](#) in Router-CIM Automation if this option is selected.

Note: On some CNC machines, the 'Tool Comment' field will be what directs the machine to a particular tool instead of 'Tool Number'. Please refer to your post processor's Application Notes to determine what your particular CNC machine will need.

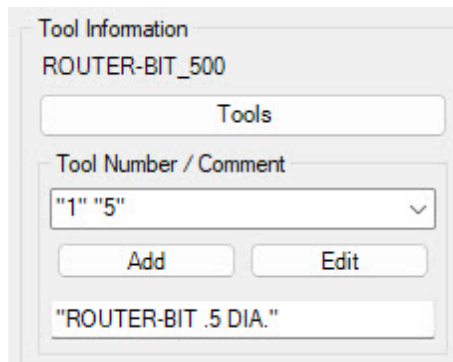
Tool Number

The main tool or spindle number, and any active spindles to use for the current cut are entered into this field as strings (in quotation marks).

Enter the main spindle as the first tool number in quotes, enter each other active spindle in quotes following the main spindle, separate with a space.

Example1: Tool Num/Comment: "1" "5" uses Main spindle #1 as the lead and spindle #5 will also be active.

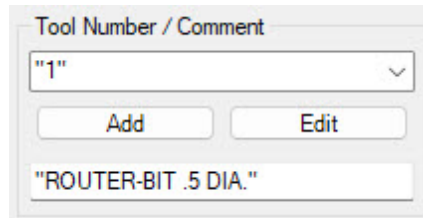
Note: There must be a space between the tool calls.



The screenshot shows a 'Tool Information' dialog box. At the top, it says 'ROUTER-BIT_500'. Below that is a 'Tools' button. The 'Tool Number / Comment' field is a dropdown menu showing '"1" "5"' with a downward arrow. Below the dropdown are 'Add' and 'Edit' buttons. At the bottom, there is a text field containing '"ROUTER-BIT .5 DIA."'.

Example 1

Example2: Tool Num/Comment: "1" uses Main spindle #1 and no other active spindles.



This is a close-up of the 'Tool Number / Comment' dropdown menu. It shows the value '"1"' selected. Below the dropdown are 'Add' and 'Edit' buttons. At the bottom, the text field contains '"ROUTER-BIT .5 DIA."'.

Example 2

Note: On some CNC machines, the 'Tool Number' field may require a different tool number call then what has been stated above. Please refer to your post processor's Application Notes to determine what your particular CNC machine will need.

Your router may use a different numbering scheme, but the tool number will be entered in the same way and in the same place. The correct tool numbers for your machine will be shown in the Application Notes for your post processor.

Tool Comment

The tool comment parameter is simply a comment field that will generate a comment in the NC Code file prior to the tool change to the designated tool.

- You can place any syntax in this field you like up to 80 characters.
- Some machine tool controllers (like Fanuc) cannot represent lower case characters. So you should use capital letters in this parameter.
- The entire string placed in this parameter must be contained in quotation marks (" ").

- This field will also represent the tool in the 'Summary Report' in Router-CIM Automation if this option is selected.

CRC Offset

CRC Offset

This location generates a D value for using Cutter Compensation (G42) or (G41) depending on the post processor. The task called autocrc will automatically use the specified tool number for the CRC Offset number only when the tool attribute Cutter Compensation is set to **'Yes'** or **'Both'**.

If Cutter Compensation is set to **'No'** this field is ignored.

When the CRC Offset field is set to autocrc, then Router-CIM will automatically generate a D number for you as long as the specified tool number is a valid number for the post and is not a drill.

CRC Offset
Automatic setting of D value

You can put any number into this field if you don't want to use the Autocrc task (if you don't want the system to automatically generate a D number). The number must be in quotes to be considered valid.

CRC Offset
Manual setting of D value

Remember that the Post processor system will automatically assign a D value for each tool as long as autocrc is listed in this location. Check the Application notes for the offset number for each tool. Replacing the number in the autocrc field with any number (in quotes) will set that number for the D value in the cut, and can also be stored in a knowledge so that you do not have to reset the number again.

Spindle Direction

This field allows you to change spindle rotation from clockwise (CW) to counter-clockwise (CCW) by placing the correct text in this box. The only valid entries are CW or CCW. Few machines support the CCW option. Check your machine's specifications to see if it can output the code necessary to start the spindle in either CW or CCW. If your machine tool does not accept the CCW command, then you will likely get an error when the NC code file is run.

Spindle Direction
Spindle Clockwise

Spindle Direction

Spindle Counter-Clockwise

Tool Diameter

Tool Diameter

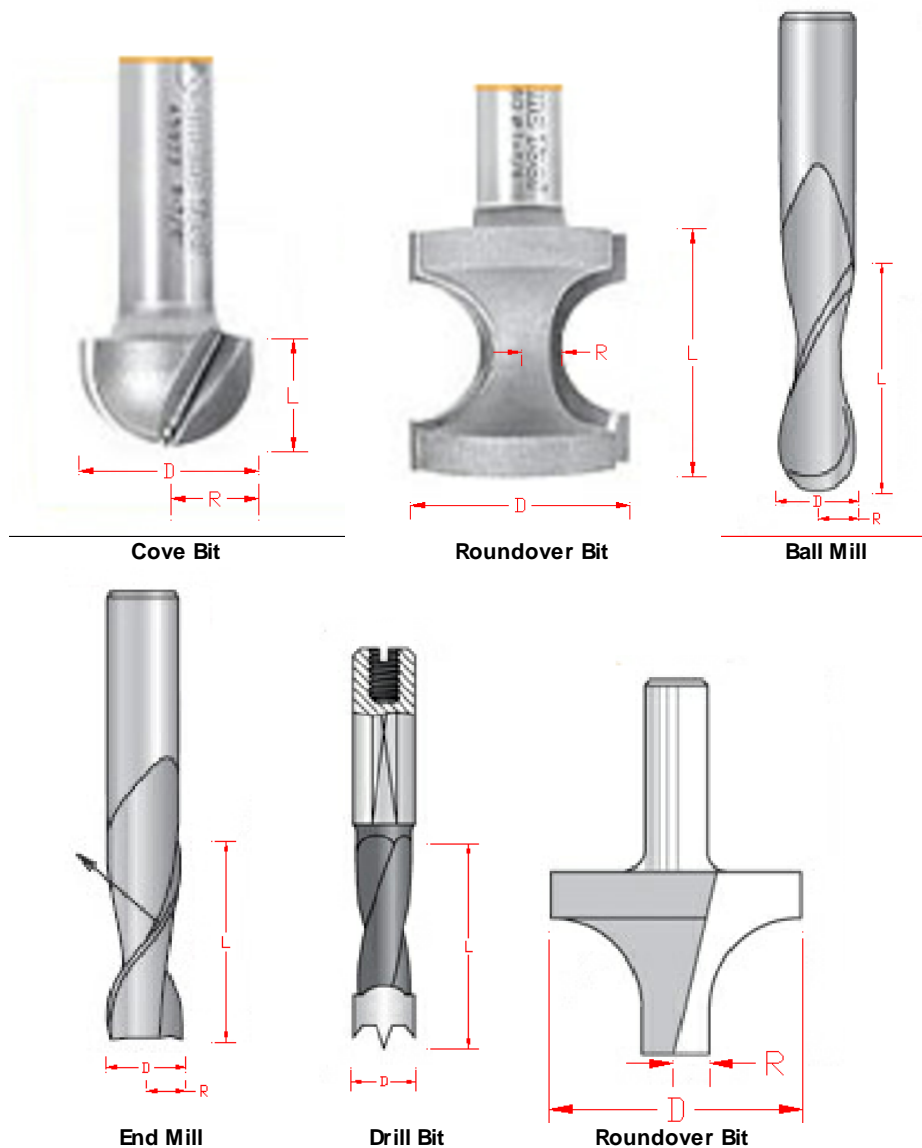
This parameter holds the Diameter of the currently selected tool.

The field can be edited if you wish to temporarily change the value of the currently selected tool. Changing this field does not change the value of the tool in the tool list! It is for temporary editing only.

This value will affect the size of the lead-in and lead-out on most cut cycles, but will not affect the offset of the tool paths.

If you are using the Cut-SIM (Tool Simulation Software), this will change the size of the tool reported to Cut-SIM and result in a back plot that is inaccurate.

The diameter value is the largest diameter value of the tool if more than 1 option is available, for instance on a shaper type tool. Some common tools and their measurements are shown here.



In each of these tools L is the Tool Length, D is the Tool Diameter, and R is the Tool Radius.

Tool Radius

Tool Radius

This parameter holds the Radius of the currently selected tool.

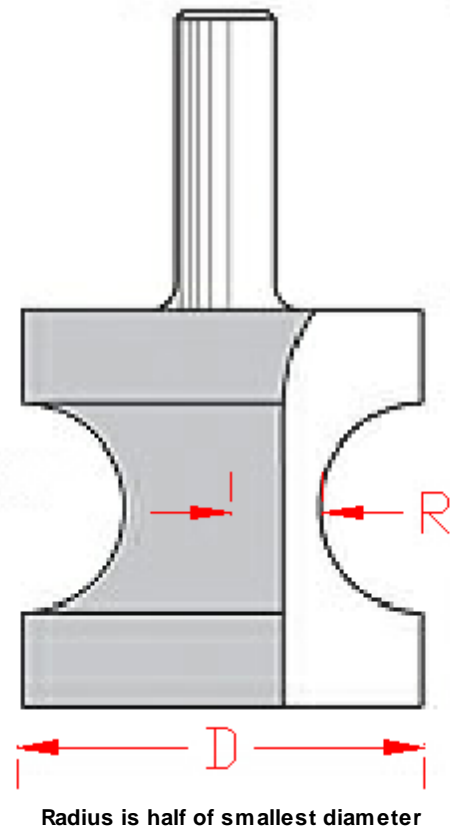
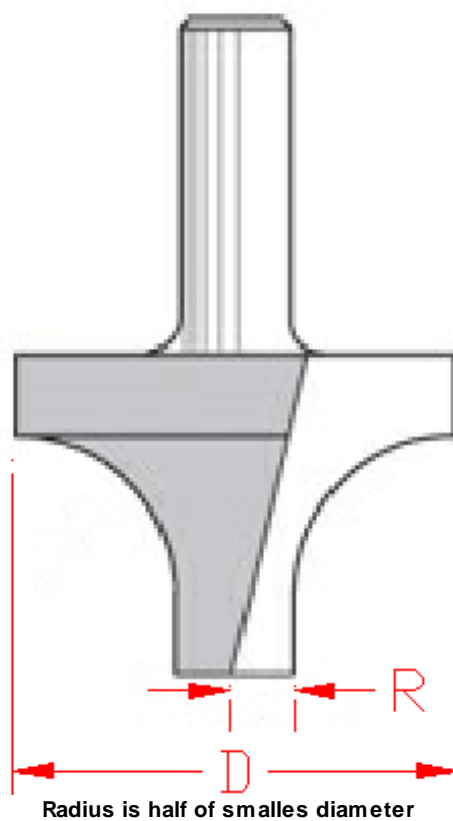
The field can be edited if you wish to temporarily change the value of the currently selected tool. Changing this field does not change the value of the tool in the tool list! It is for temporary editing only. This value will determine the tool path offset distance from the geometry when Cutter Compensation is set to **'No'** or **'Both'**.

If you are using the Cut-SIM (Tool Simulation Software), this will change the size of the tool reported to Cut-SIM and result in a back plot that is inaccurate.

You should instead use the [Offset Dim](#) or the [XY Stock Allowance](#) under cycle information to change the amount of the tool offset.

When setting the tool radius, it is important to remember that this is used as the offset for the tool path and if you are using a shaper cutter, the radius may not be half of the diameter.

Look at the following images:



In both of these cases you would only want to offset the tool path by half of the smallest diameter. However for lead-in and lead-out clearance, the diameter is the largest diameter so that you account for the entire size of the tool in the tool path. In this case the radius is not half of the diameter.

Tool Length

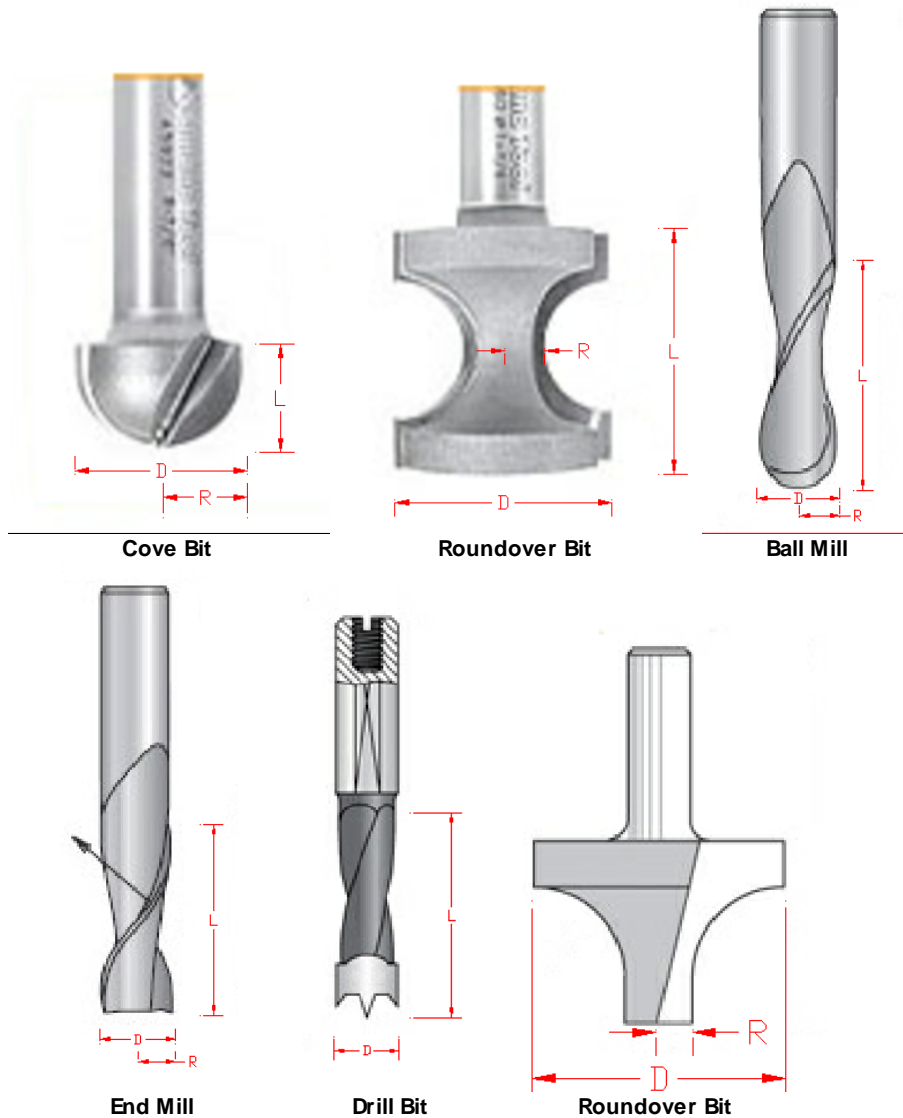
Tool Length	4.00000
-------------	---------

The Tool Length field is used to set the length of cutting flute on a tool. It is also the maximum depth that Router-CIM will allow this tool to move in Z.

For temporary editing, you may change this value on the Control Panel.

When setting this value, you should pay some attention to how deep each tool can cut, especially if you are going to make a cut at full depth in one pass.

A typical setting of tool length for common tools would be the L setting in the following examples.



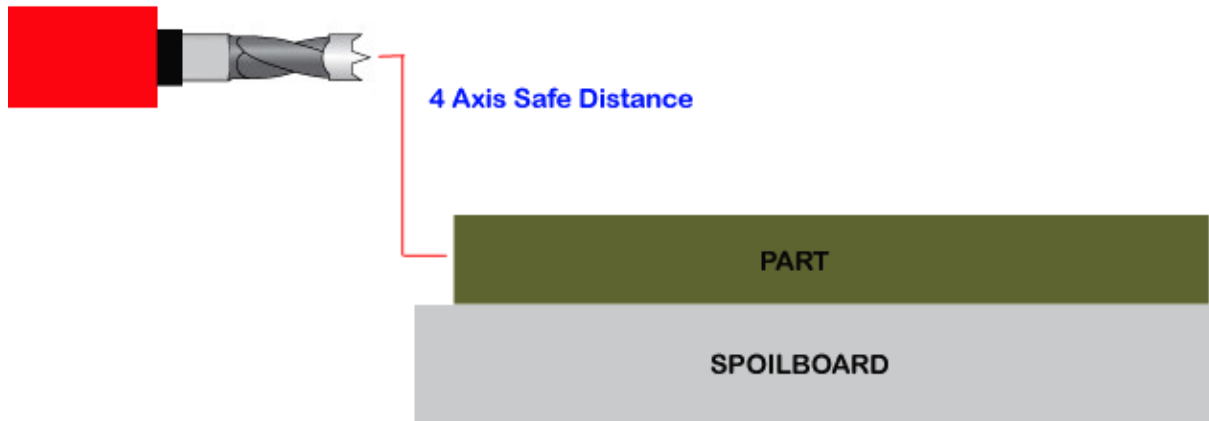
In each of these tools L is the Tool Length, D is the Tool Diameter, and R is the Tool Radius.

4 Axis Safe

4 Axis Safe

4.00000

This field allows for a safety retract height for a horizontal tool such as a horizontal boring head. This value will show the height to which the tool will return after each Cut, and also the height it will traverse at between cuts.



This parameter is used for horizontal drilling and for horizontal machining only. If no value is input, and one is necessary, then you will be prompted on the screen for the 4 Axis Safe amount at the command line.

Positive values only are expected.

Tool Type and Tool Category

Tool Type	<input type="text"/>
Category	<input type="text"/>

Type and Category are used for certain types of CNC machines. Currently these fields are used by the system, and no user input is needed. Your Router-CIM Post Processor's Application notes will give you further information on these fields if necessary.

Tool Type for Engraving Tool

When an Engraving tool (VBIT) is selected, the tool type field will allow you to build a tool for the CutSIM add-on and for the Advanced Cycles add-on.

Tool Information
ENGRAVE_V-BIT

Tools

Tool Number / Comment
"1" ✓
"V BIT FOR ENGRAVING"

CRC Offset autocrc

Spindle Direction CW

Tool Diameter 0.06250

Tool Radius 0.03125

Tool Length 4.00000

4 Axis Safe

Tool Type Engrave Tool

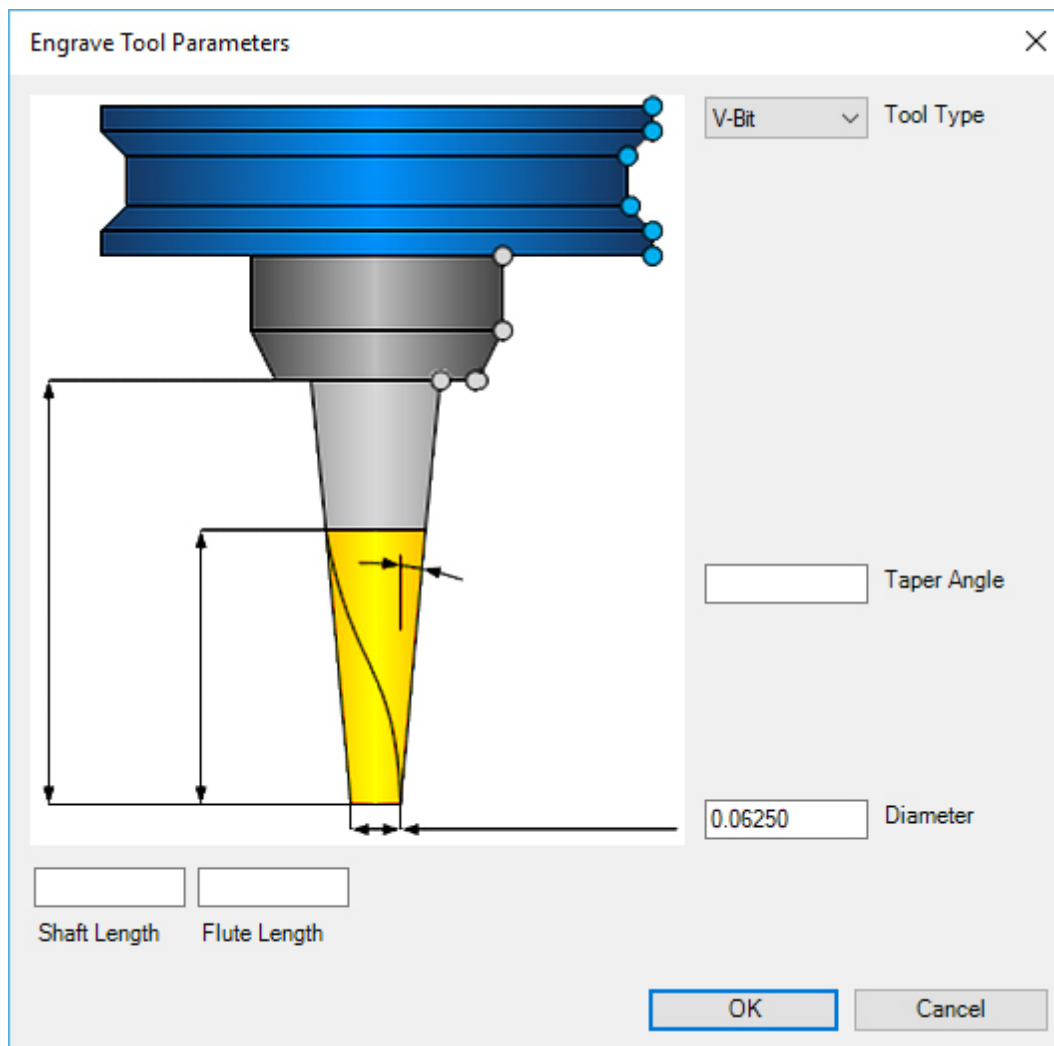
Category

Vertical Offset

Horizontal Offset

Cutter Bridge

When the 'Engrave Tool' button is selected, the 'Engrave Tool Parameters' screen will open.



To define an engraving (VBIT), you will need to fill in the following fields:

- 1) Shaft Length - Distance from tip of the tool to maximum effective depth of the tool
- 2) Flute Length - Distance from tip of the tool to flute/cutting edge maximum
- 3) Taper Angle - Angle defined by the center to taper
- 4) Diameter - Distance from minimum tip of the tool. The smallest value acceptable in this field is 0.002

Vertical and Horizontal Offset

Vertical Offset	<input type="text"/>
Horizontal Offset	<input type="text"/>

The Vertical and Horizontal Offsets are used for certain types of CNC machines. Currently these fields are used by the system, and no user input is needed. Your Router-CIM Post Processor's Application Notes will give you further information on these fields if necessary.

Cutter Bridge (Multi-Bridge)



When a knowledge has this field populated with a positive number, it will create an additional cutter bridge (offset) when it is nested through Router-CIM Automation Suite if the knowledge is used.

When this knowledge(s) is used, the outside shaped defined by the nesting engine will be increased by the value in the Cutter Bridge field.

The most common situations to use this feature are for small parts where you want an additional cutter bridge (offset) in order to better hold the part or if you are using a profile cutter on specific parts and you need the additional cutter bridge (offset) in order to prevent the profile tool from interfering with parts that may be nested next to it.

Note: Only one knowledge with a Cutter Bridge value may be used per part. Multiple knowledges with a Cutter Bridge value may be used in a job as long as they are assigned to different layers.

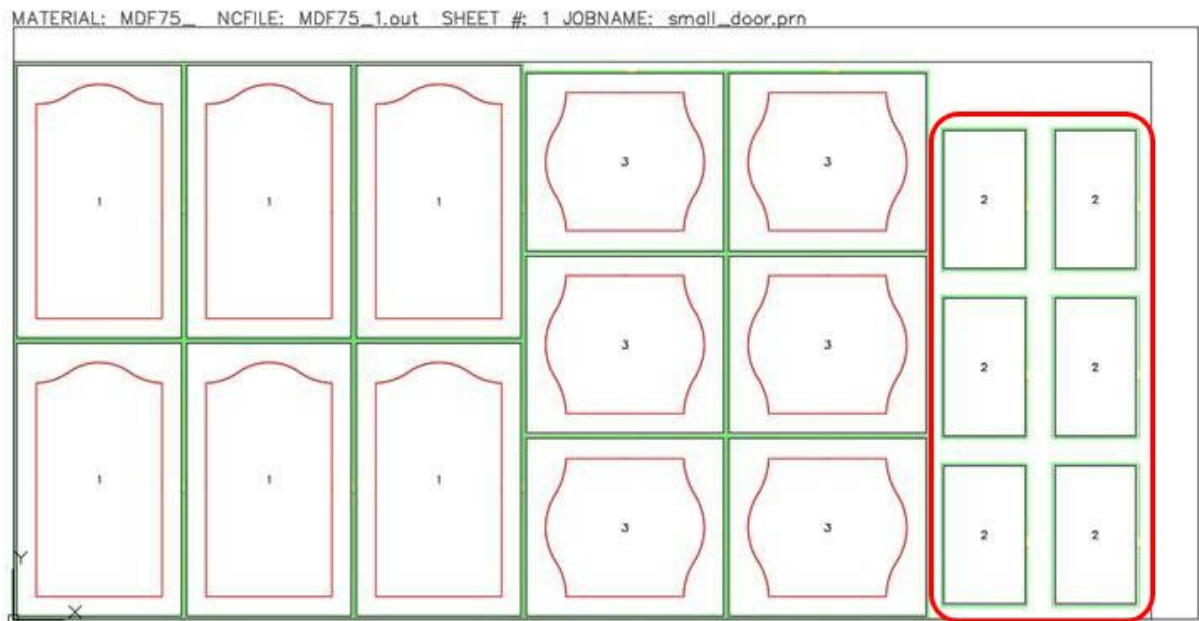
For example, if you need to have an additional cutter bridge (offset) around a part that has a specific profile that the normal bridge width set for that material will not be able to accommodate, you can use the Cutter Bridge field to add the additional offset needed.

The image shows a 'Tool Information' dialog box for 'ROUTER-BIT_500'. It contains various input fields for tool parameters. The 'Cutter Bridge' field, located near the bottom of the list of fields, is highlighted with a red rectangular border. The value '.5' is entered in this field. Other fields include 'Tool Num / Comment' (with sub-fields for '1' and 'RBIT 500 DIA. COMP'), 'CRC Offset' (autocrc), 'Spindle Dir' (CW), 'Tool Dia.' (.5), 'Tool Radius' (.25), 'Tool Length' (1.375), '4 Axis Safe', 'Type', 'Category', 'Vert. Offset', 'Horz. Offset', 'Aggregate Offset' (with 'Collet' selected), and 'Cutter Compensation' (with 'Both' selected).

Tool Information	
ROUTER-BIT_500	
Tools	
Tool Num / Comment	
"1"	<input type="checkbox"/>
"RBIT 500 DIA. COMP"	
CRC Offset	autocrc
Spindle Dir	CW
Tool Dia.	.5
Tool Radius	.25
Tool Length	1.375
4 Axis Safe	
Type	
Category	
Vert. Offset	
Horz. Offset	
Cutter Bridge	.5
Aggregate Offset	
<input type="radio"/> Spindle	<input checked="" type="radio"/> Collet
Cutter Compensation	
<input type="radio"/> Yes	<input type="radio"/> No <input checked="" type="radio"/> Both

Note: In order for Router-CIM to use this field, the 'Enable Cutter Bridge' option needs to be checked in the 'Small Part Options' area under the [Job Settings](#) tab in Router-CIM Automation Suite

When this knowledge is used during a job run in Router-CIM Automation Suite, an additional bridge will be added to the boundary of the part so there will be a larger overall bridge width when the part is nested by Router-CIM Automation Suite:



Aggregate Offset



This parameter is the offset type of a horizontal tool path and applies only when making Horizontal Cuts.

This type of offset is dependent on the settings that are available to the programmer/operator when programming and running the machine. The control of the cut and tools can be solely placed in the program to accomplish all the machine offset moves or it can be placed in the machine, or a combination of both.

Spindle

If Spindle is chosen as the aggregate offset, the cut path will appear on the screen to be offset by the amount designated in the Tool Length parameter.

Collet

If Collet is chosen, the cut path will appear inside the part the distance designated in the Total Depth parameter on the Control Panel. This method requires that you either set the work coordinate to the tip of the tool, or set the distance from the tip of the tool to the center of the spindle in the horizontal tool length offset and use Plane Detect.

The Combination of Spindle and Collet methods

There is a third choice for using a horizontal drill. It is probably the easiest for both the programmer and operator alike. This method requires that you are cutting on one of the 4 faces of the part and not some arbitrary angle, and also for the programmer to know the distance from the center of the drill block to the face of the collet that the tool fits into. It then requires the operator to measure how much tool is sticking out of the collet in the drill block.

The programmer places the distance from the center of the spindle (where the work coordinate is set) to the face of the collet in the Tool Length.

The operator placed the tool length (the amount of the tool sticking out of the collet) into the horizontal length offset.

This method allows for the programmer to only need one number that never changes for a tool length, and the operator can measure the amount from the collet to the tip of the tool (with almost any measuring device) and put it in an offset, just like he/she would for any other tool touched off.

Spindle Offset

Spindle

If Spindle is chosen as the aggregate offset, the cut path will appear on the screen to be offset by the amount designated in the Tool Length parameter.

The Plane Detect is not necessary, but if it is used, then the offsets will be set to 0.

For an example of how this method would work, the following explanation is offered.

Looking at the picture below, the tool is a 3/8" Drill Bit, in a horizontal drill spindle.

The 4th axis Safe Plane is set to 2.0.

The Tool Length is set to 4.25.

The Aggregate Offset is set to Spindle.

The Safety Plane is *.25, and the Cut Depth is -1.0.

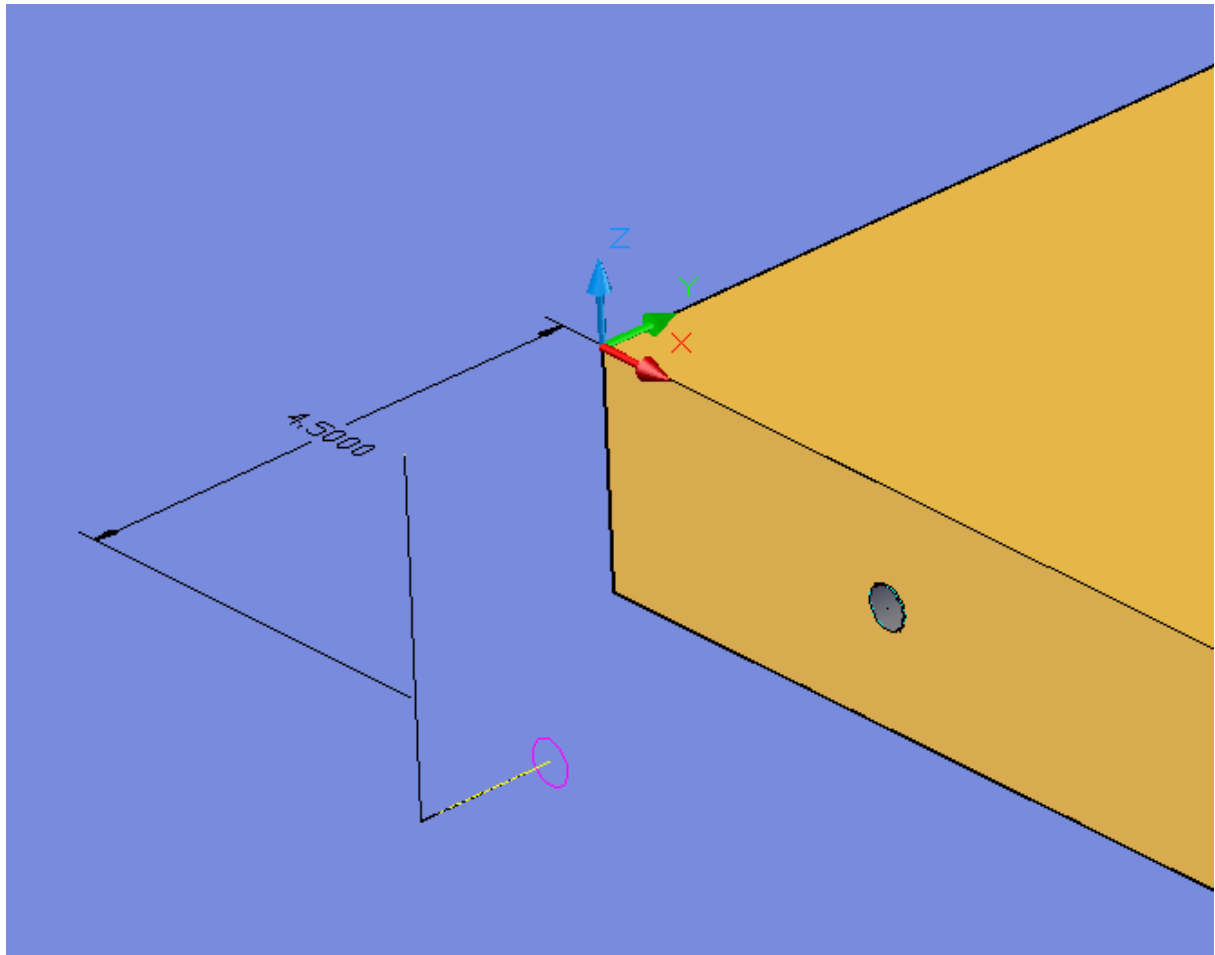
The screenshot displays the CIM-Tech Router-CIM Control Panel interface. Key settings are highlighted with red boxes:

- Tool Information:**
 - Tool Number / Comment: "1003"
 - Tool Description: "HORIZONTAL DRILL .375 DIA."
 - Tool Length: 4.25000
 - 4 Axis Safe: 2.00000
 - Aggregate Offset: ☒ Spindle
- Status Information:**
 - Safety Plane: 0.25000
 - Depth Per Pass: 1.00000
 - Total Cut Depth: -1.00000
- Knowledge / Settings:**
 - Tabbing: ☒ No
 - Acc-n-Dec: ☒
 - Plane Detect: ☐

The interface also includes a central area for cycle information, a bottom toolbar with icons for Geoshape, Group, Start Point, Cut, Sequence, Seq. Undo, Edit Code, Mod Cycle, Mod Tool, and NCVars, and a status bar with OK and Close buttons.

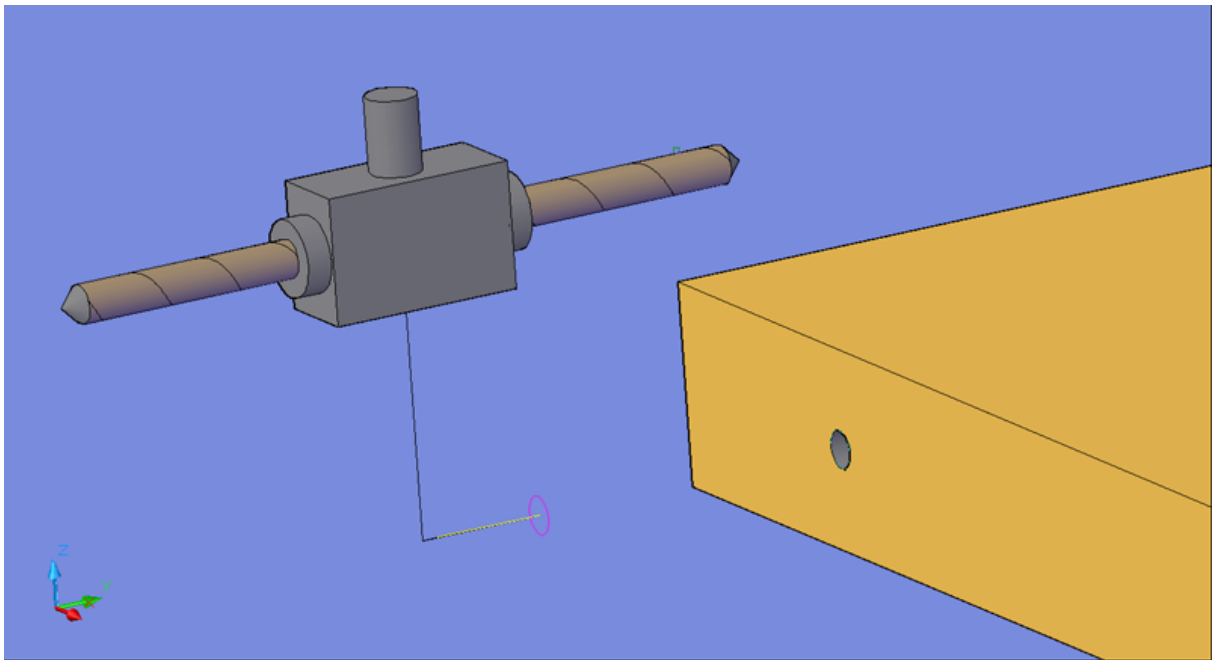
Horizontal Drill Example.

The effect of this is that the Tool Path will be offset from the geoshape by 4.5". This dimension is 4.25 for the tool length PLUS .25 for the Safety Plane.



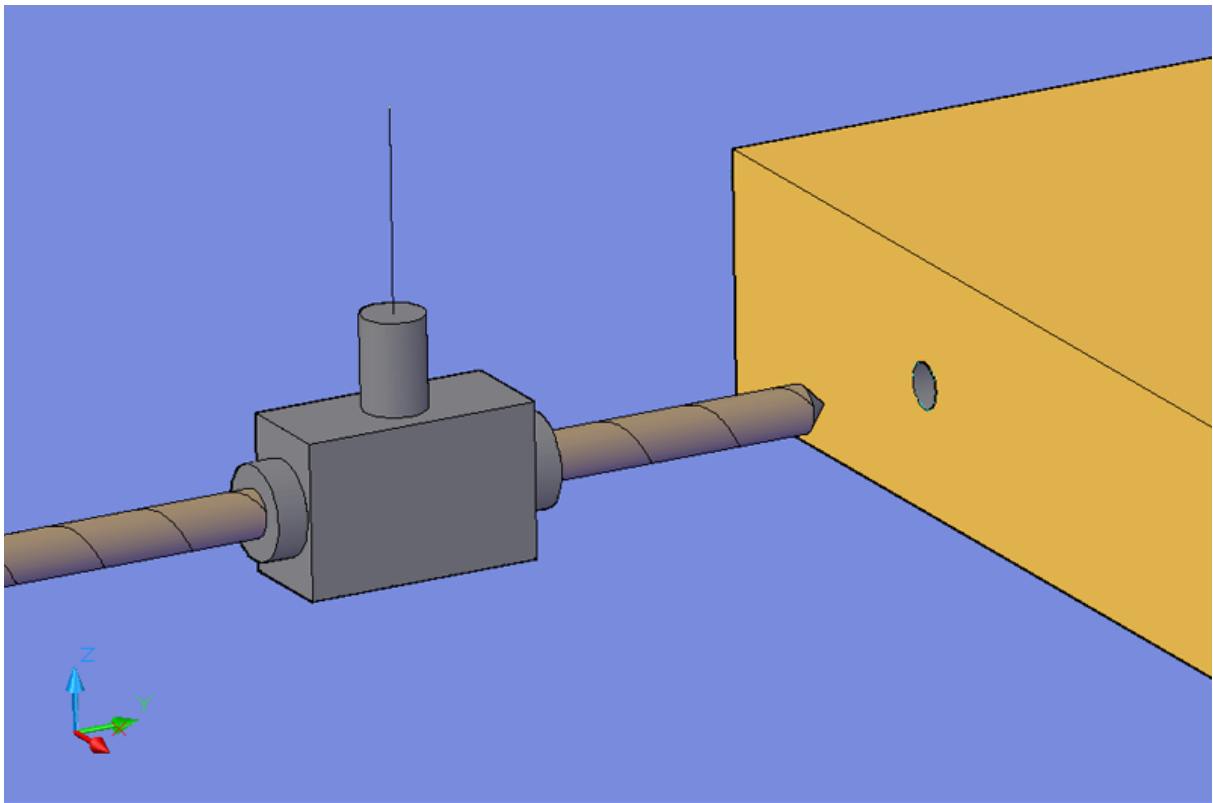
Tool Path offset by Tool Length + Safety Plane.

The tool path will represent the center of the drill block in this example. That means that the center of the drill block will move to the end of this tool path which starts at Z2.0 (4th Axis Safe) and the XY position 4.5 inches from the face of the drill hole.



Move to 4th Axis Safe

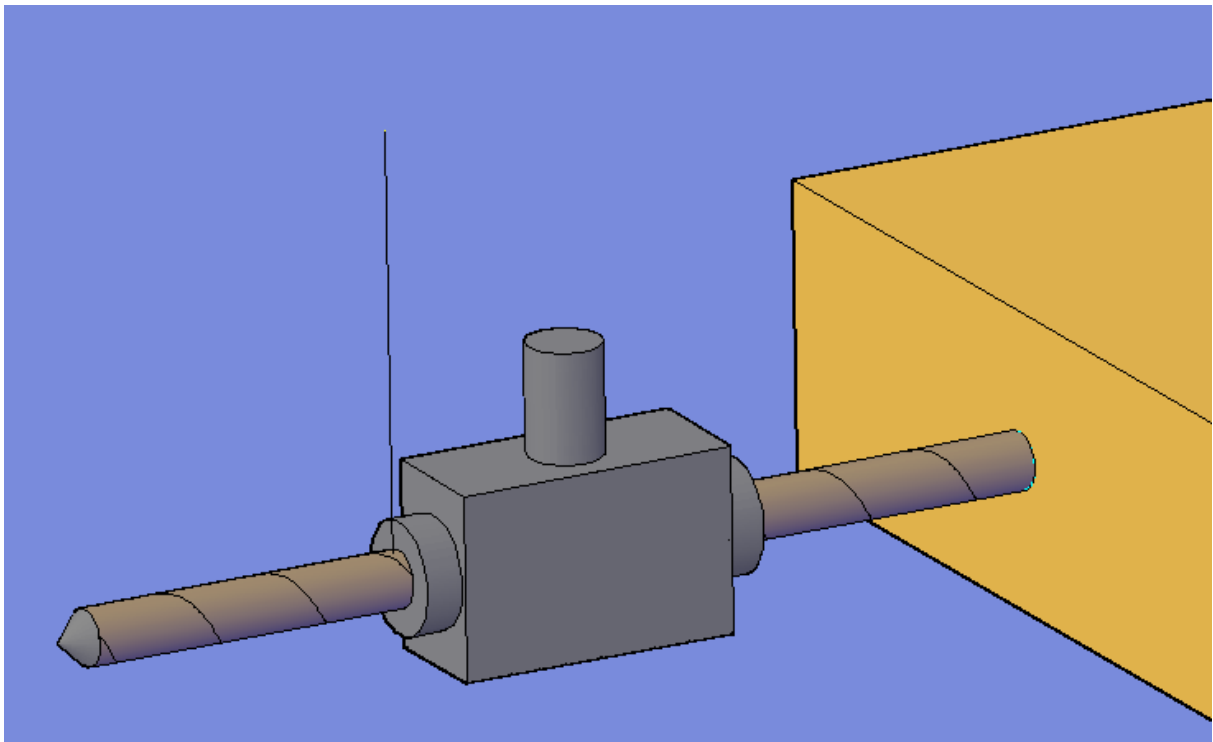
Next the tool will move straight down to the center of the drill hole in Z. It will still be .25 (Safety Plane) away from the center of the hole. This is the small black section of the tool path shown above. The tool will move from that .25 away from the hole, drill all the way to the bottom of the hole (1" into the material) and then move back out to the point 4.5 inches away from the center of the hole, then lift back up to the 4th Axis Safe point (Z2.0).



Move down to center of the hole, .25 away from face.

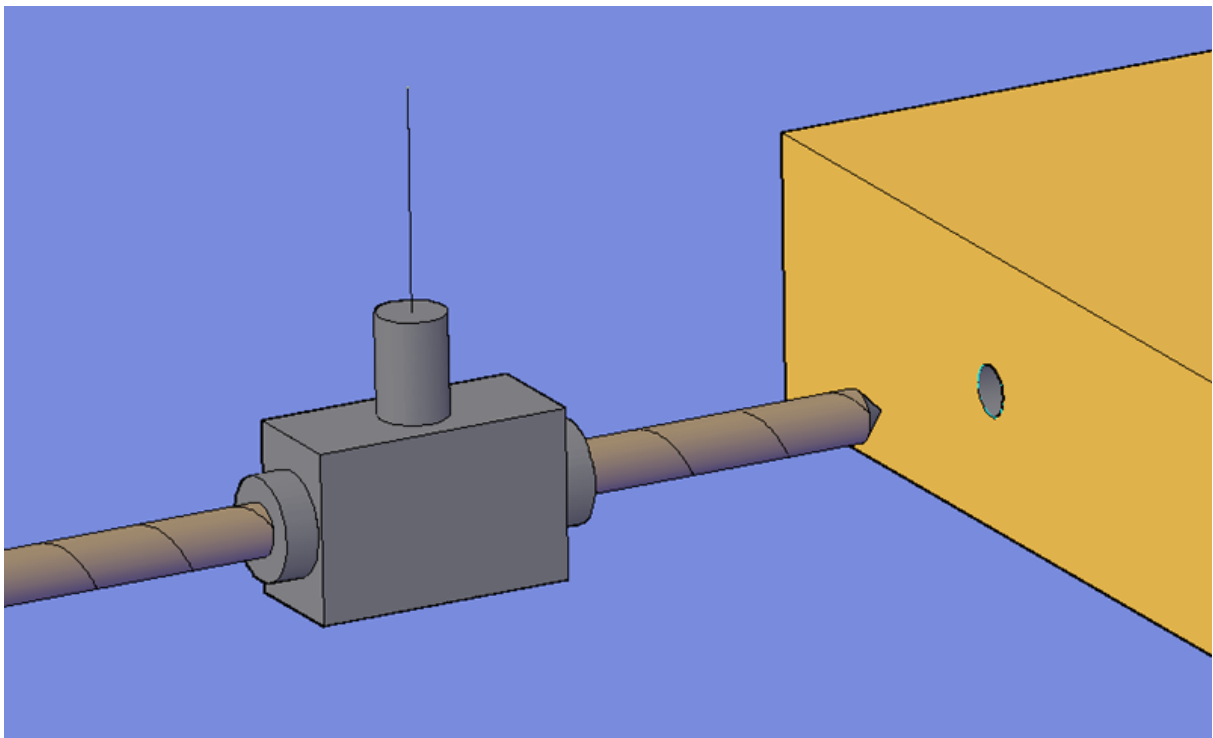
Here is the move down in Z to the center of the hole, still .25" away from the face of the part. This is a Rapid Traverse (G0) move.

Next the hole will be drilled 1.0" deep...



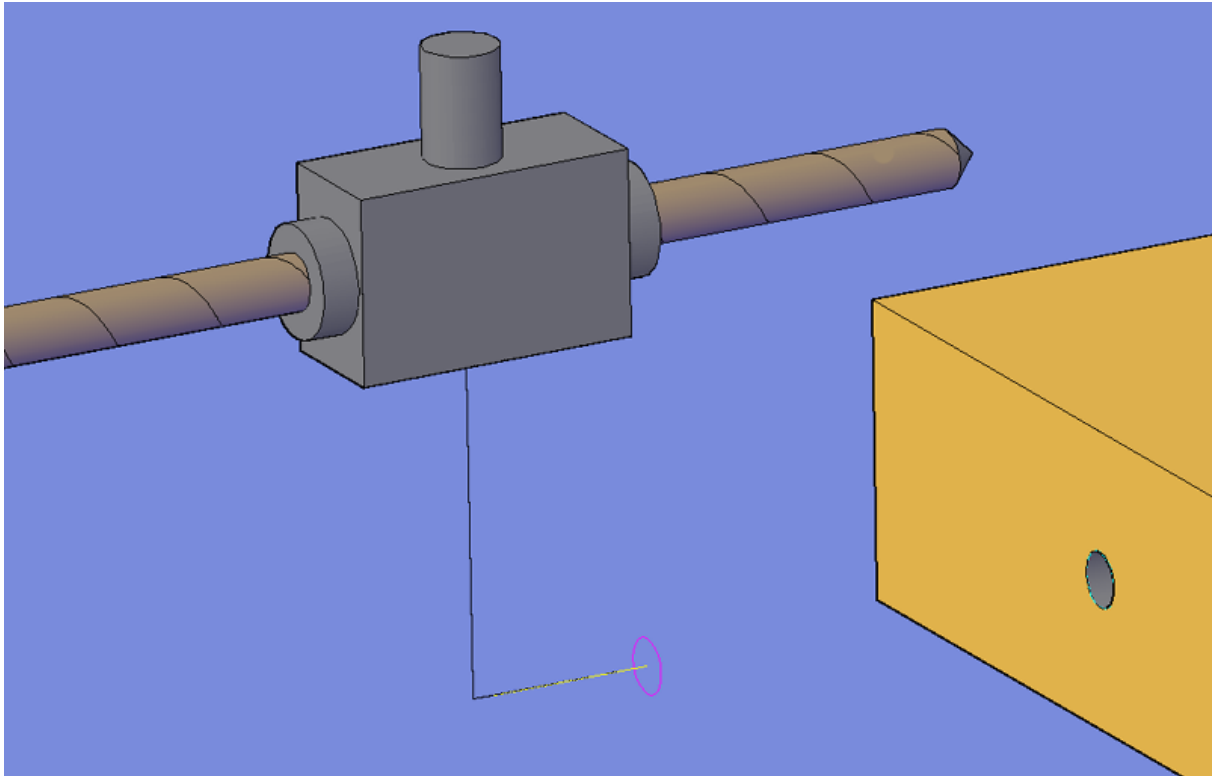
Drill the hole.

Drill the hole 1.0" deep. This is a feed move (G1) at the programmed feedrate.
Next the tool will retract back to the Safety Plane.



Retract to the Safety Plane.

The tool retracts to the Safety Plane. This is a Rapid Traverse (G0) move. Next the tool will raise back up in Z to the 4th Axis Safe.



Tool retracted to 4th Axis Safe.

And the tool is now retracted to the 4th Axis Safe. This is also a Rapid Traverse (G0) move.

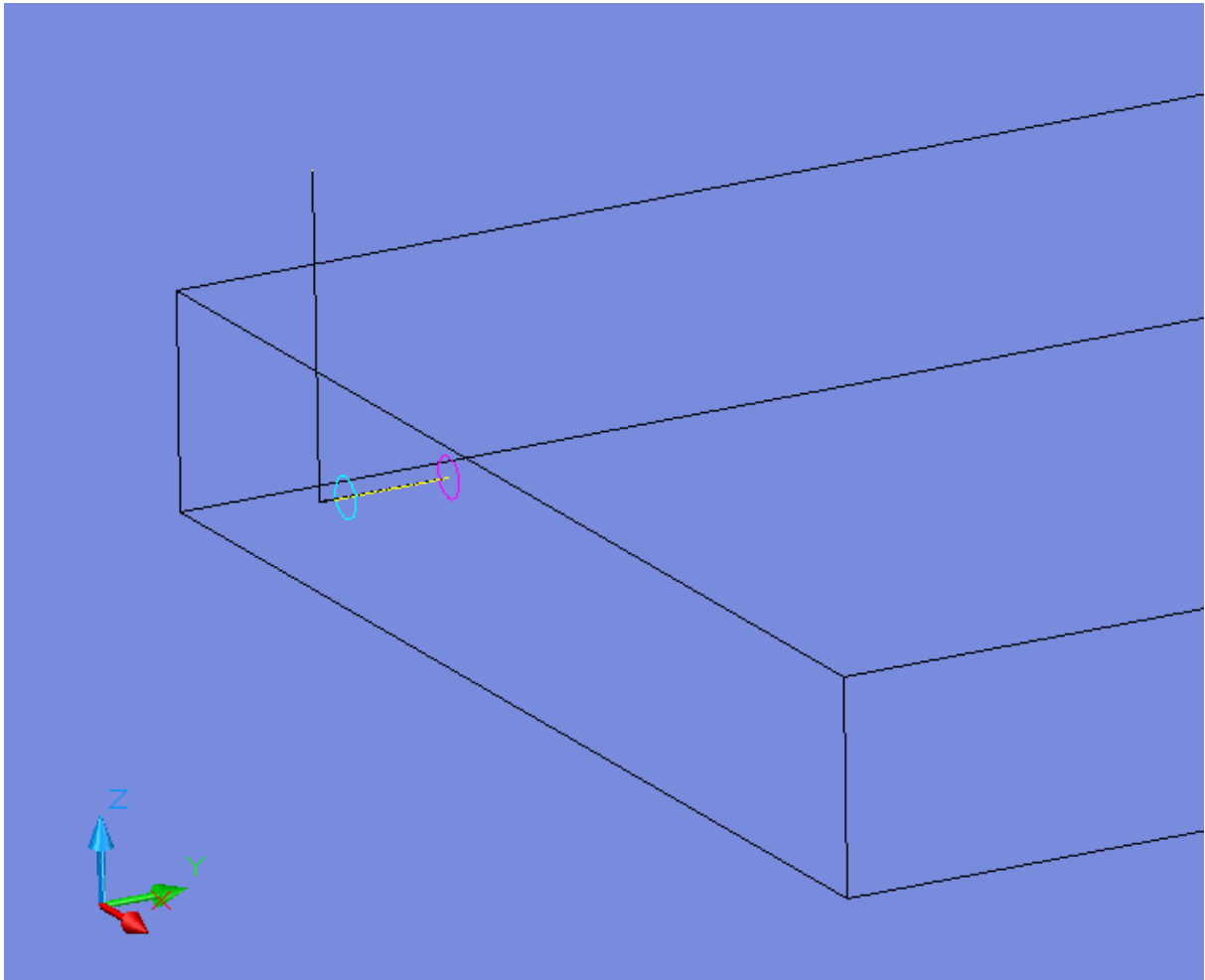
This method only require that the programmer have an idea of the distance from the center of the spindle to the tip of the drill bit. That is an important measurement, but is really the only one required.

In the picture above, the tool path has been shifted by the value in the tool length field to emulate the spindle center, where the work coordinate or origin is set.

Collet Offset

Collet

If Collet is chosen, the cut path will appear inside the part the distance designated in the Total Depth parameter on the Control Panel. This method requires that you either set the work coordinate to the tip of the tool, or set the distance from the tip of the tool to the center of the spindle in the horizontal tool length offset and use Plane Detect.



Horizontal Drill with Collet Offset tool path.

For an example of how this method would work, the following explanation is offered.

Looking at the picture below, the tool is a 3/8" Drill Bit, in a horizontal drill spindle.

The 4th axis Safe Plane is set to 2.0.

The Tool Length is set to 4.25.

The Aggregate Offset is set to Collet.

The Safety Plane is *.25, and the Cut Depth is -1.0.

CIM-Tech Router-CIM Control Panel

Description

Tool Information
DRILL-BIT_375

Tools

Tool Number / Comment
"1003"
"HORIZONTAL DRILL .375 DIA."

CRC Offset
"1"

Spindle Direction
CW

Tool Diameter
0.37500

Tool Radius
0.18750

Tool Length
4.25000

4 Axis Safe
2.00000

Tool Type

Category

Vertical Offset

Horizontal Offset

Cutter Bridge

Aggregate Offset
☐ Spindle ☒ Collet

Cutter Compensation
☐ Yes ☒ No ☐ Both

Cycle Information

Status Information

Safety Plane
0.25000

Depth Per Pass
1.00000

Total Cut Depth
-1.00000

Feederate/Spindle Speed

Feederate
100.00000

Spindle Speed
7500.00000

Surface FPM
NONE

Units Per Revolution
NONE

Calculate

Before Codes

After Codes

Oscillation Amount
0.00000

Sort By Rank #

Knowledge / Settings

Knowledge
Select Knowledge
CURRENT

DOIT Edit Edit

Retrieve Save

Import Export

Tabbing
☒ No
☐ Yes
☐ Auto
☐ Tab At Start
☐ Tab By Distance

Tabbing Parameters
Quantity NONE
Length NONE
Height NONE
Distance NONE
Minimum Radius 0.00000

☒ Acc-n-Dec
☐ Metric
☐ Inline
☒ Plane Detect

DOIT File
dotinfo.dat

Geoshape Group Start Point Cut Sequence Seq. Undo Edit Code Mod Cycle Mod Tool NCVars

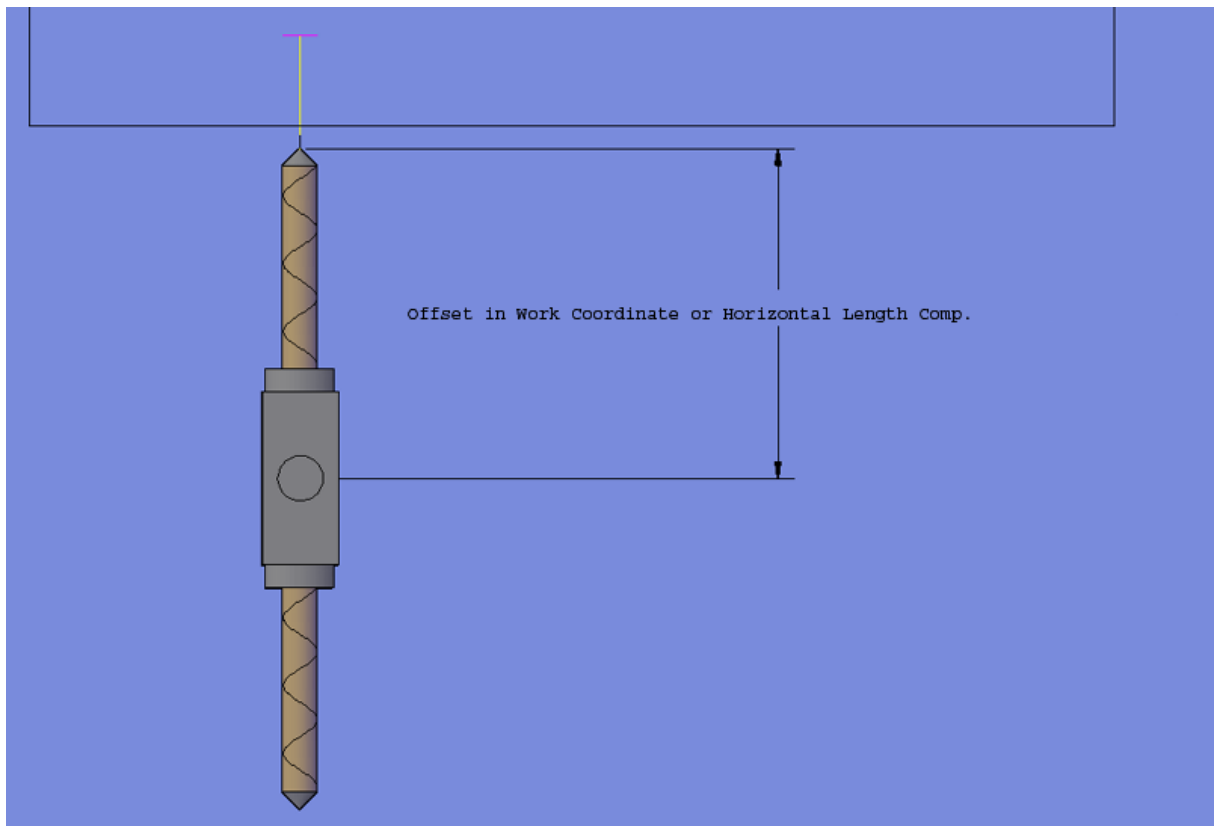
OK Close

DRILL MOTIONS

Reset Cycle Settings to Default

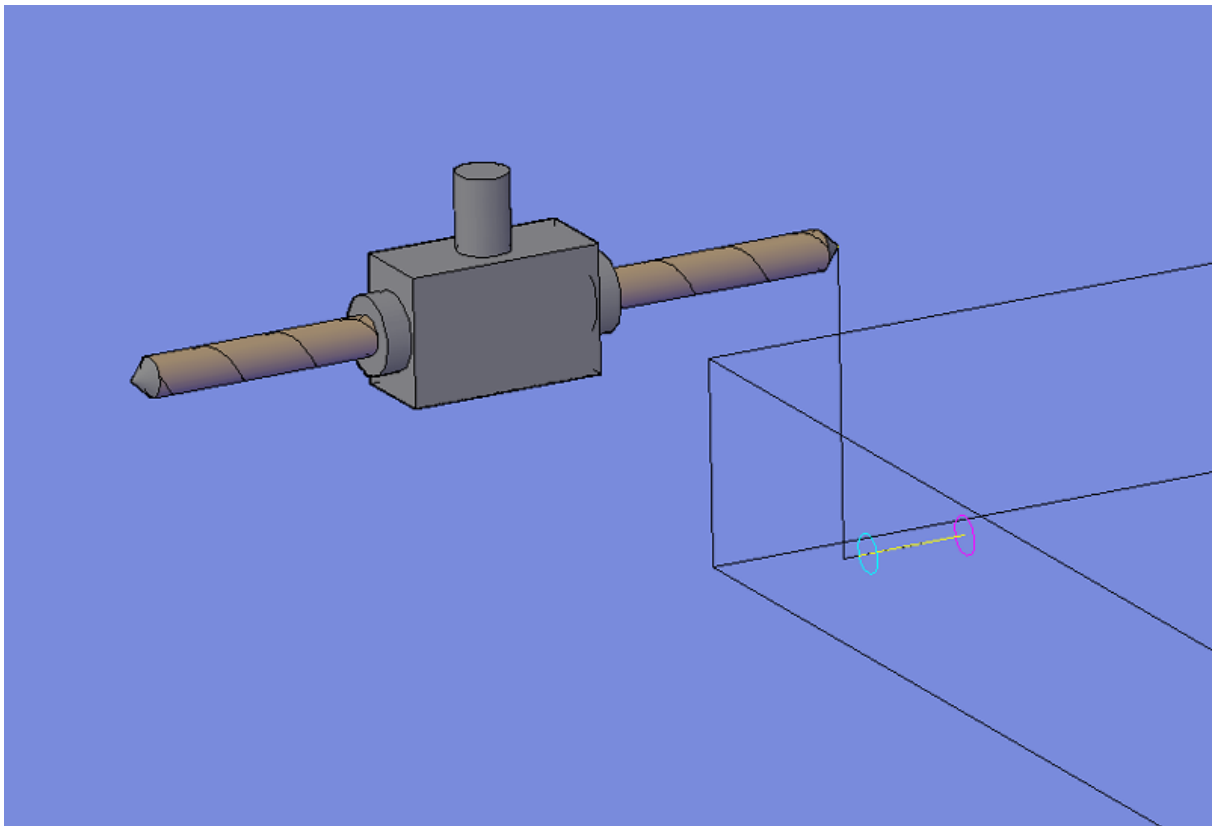
Horizontal Drill with Collet Offset

The only caveat to this system is that you must set the work coordinate to the tip of the tool, or horizontal length comp offset to the distance from the center of the drill or aggregate to the tip of the tool.

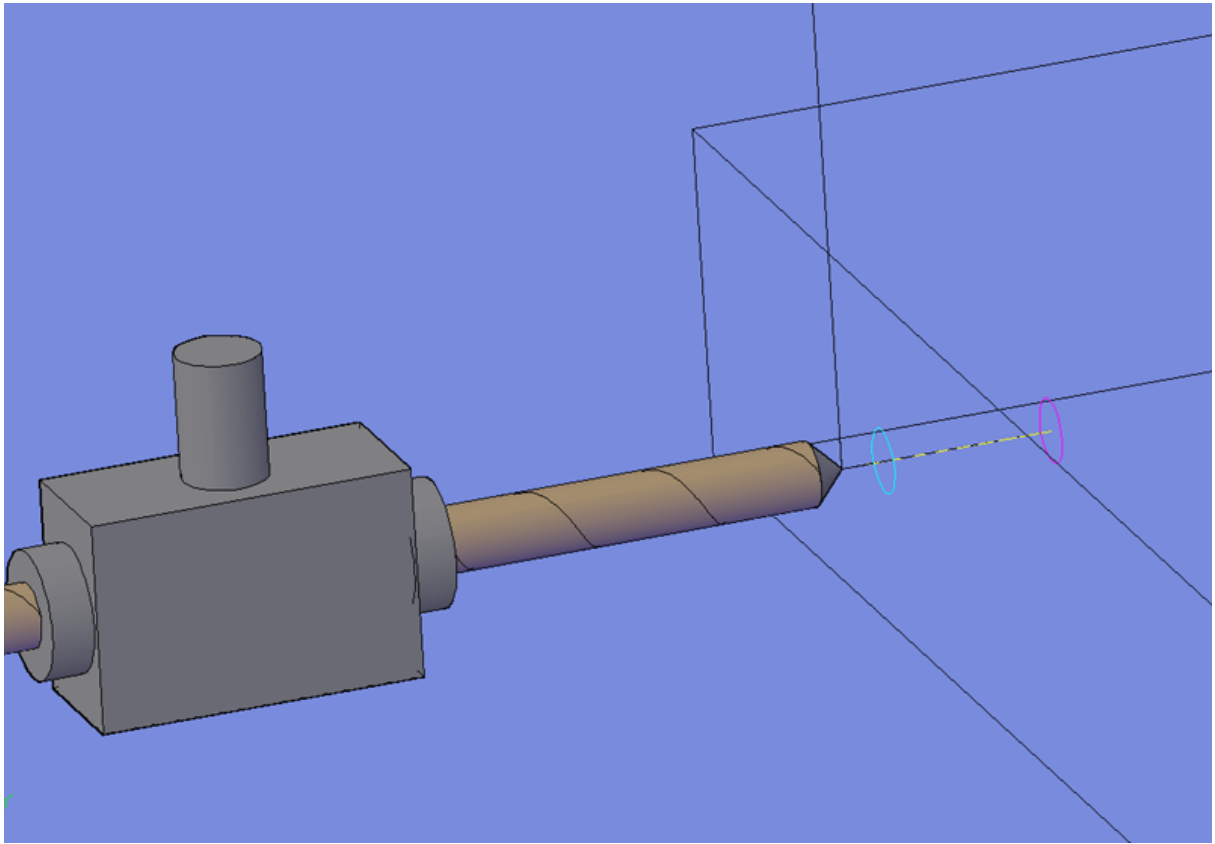


Set Distance from tip of tool to center of spindle in work coordinate or horizontal length offset.

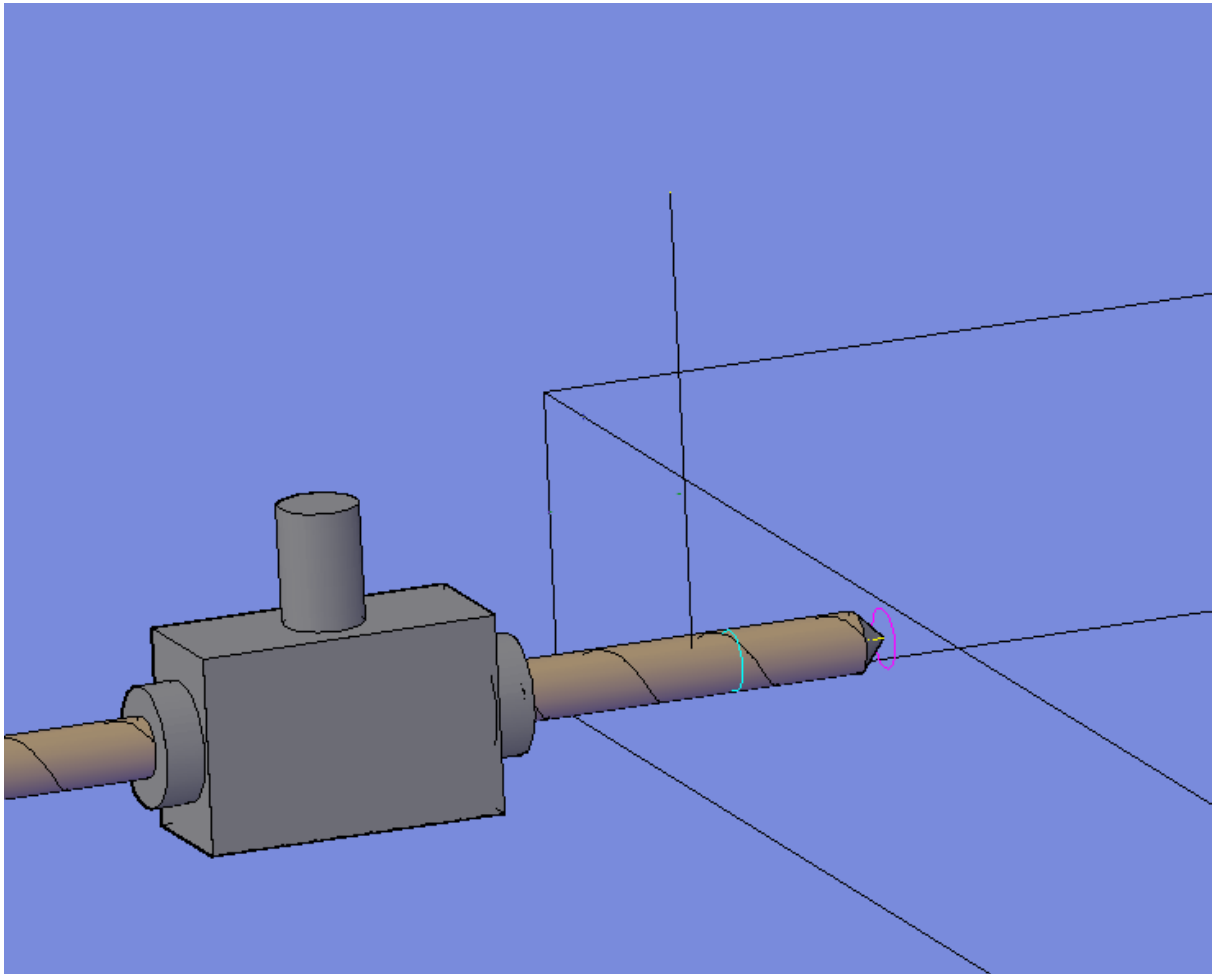
The tool will start above the part at the 4th Axis Safe point.



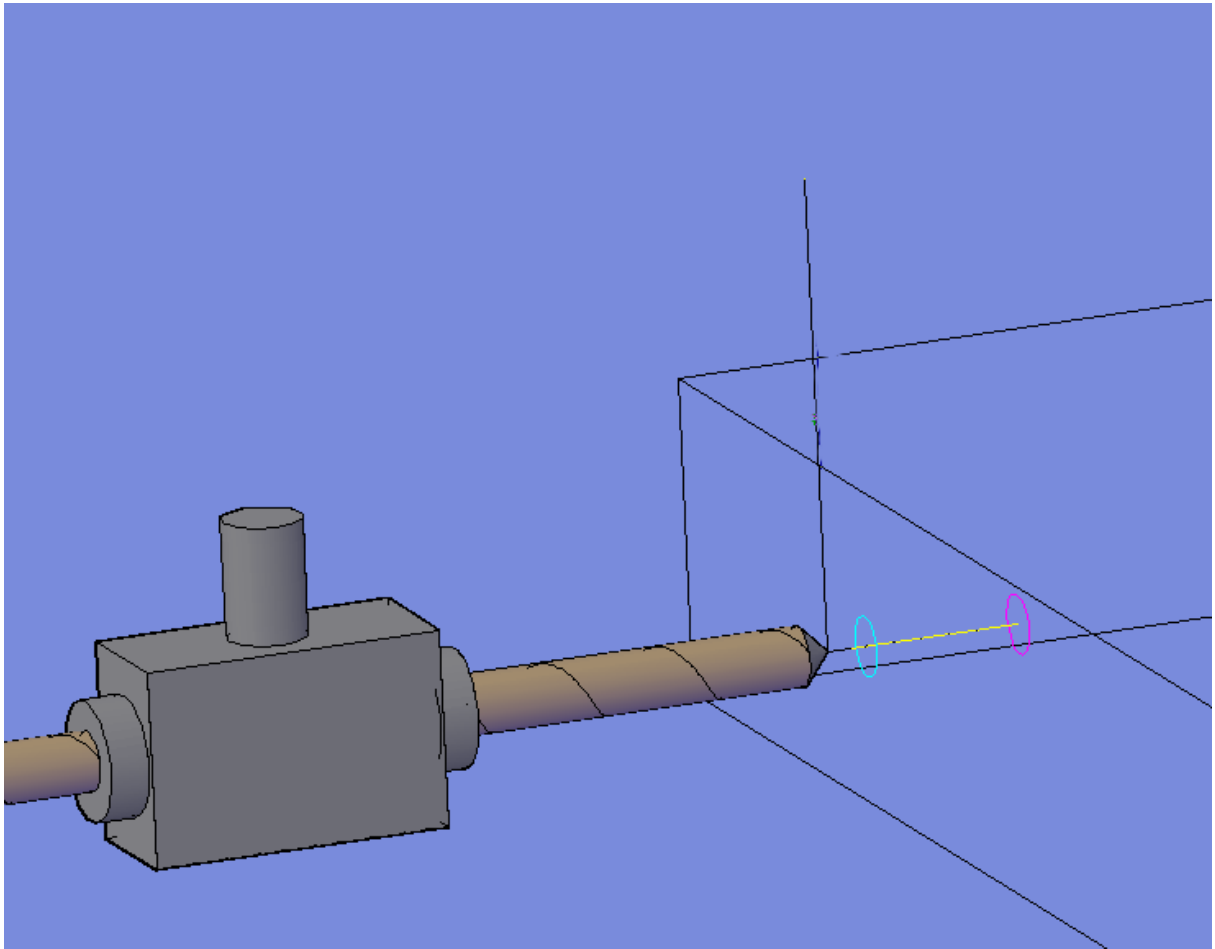
Next the tool will move down to the center of the hole, but still .25" away from the part as set by the Safety Plane. This will be a Rapid Traverse (G0) move.



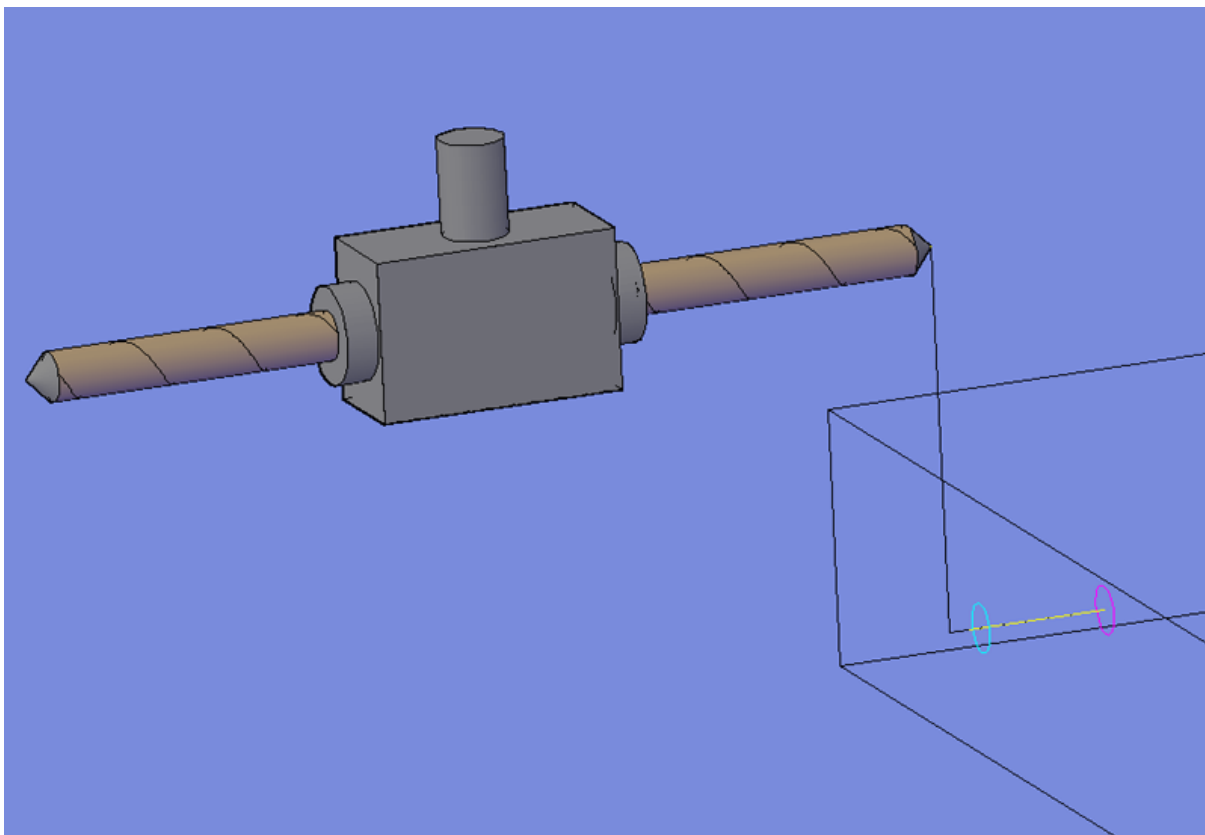
Next, the tool will feed into the part from the Safety Plane to the depth of cut (or depth of first pass if set to multiple passes) in feed mode (G1).



Once the tool has reached the bottom of the hole, it will retract back to the Safety Plane. This will be a Rapid Traverse move (G0) since there is no material to remove and the tool is retracting back to the point it just came from.



The last move will be to retract back to the 4th Axis Safe point. This will also be a Rapid Traverse (G0) move. This is the position the tool started from.



Using this method of offset only requires you to use either the Plane Detect with a Horizontal Tool Length offset or set the Work Coordinate to the tip of the tool.

Combination of Spindle AND Collet Offset

There is a third choice for using a horizontal drill. It is probably the easiest for both the programmer and operator alike. This method requires that you are cutting on one of the 4 faces of the part and not some arbitrary angle, and also for the programmer to know the distance from the center of the drill block to the face of the collet that the tool fits into. It then requires the operator to measure how much tool is sticking out of the collet in the drill block.

The programmer places the distance from the center of the spindle (where the work coordinate is set) to the face of the collet in the Tool Length.

The operator placed the tool length (the amount of the tool sticking out of the collet) into the horizontal length offset.

This method allows for the programmer to only need one number that never changes for a tool length, and the operator can measure the amount from the collet to the tip of the tool (with almost any measuring device) and put it in an offset, just like he/she would for any other tool touched off.

In Router-CIM, the knowledge would be set to Spindle, Plane Detect would be ON, and the setup would look like this for example:

CIM-Tech Router-CIM Control Panel

Description

Tool Information
DRILL-BIT_375

Tools

Tool Number / Comment
"1003"
"HORIZONTAL DRILL .375 DIA."

CRC Offset
"1"

Spindle Direction
CW

Tool Diameter
0.37500

Tool Radius
0.18750

Tool Length
2.25000

4 Axis Safe
2.00000

Tool Type

Category

Vertical Offset

Horizontal Offset

Cutter Bridge

Aggregate Offset
☒ Spindle ☐ Collet

Cutter Compensation
☐ Yes ☒ No ☐ Both

Cycle Information

Status Information

Safety Plane
0.25000

Depth Per Pass
1.00000

Total Cut Depth
-1.00000

Feedrate/Spindle Speed

Feedrate
100.00000

Spindle Speed
7500.00000

Surface FPM
NONE

Units Per Revolution
NONE

Calculate

Before Codes

After Codes

Oscillation Amount
0.00000

Sort By Rank #

DRILL MOTIONS

Reset Cycle Settings to Default

Knowledge / Settings

Knowledge
Select Knowledge
CURRENT

DOIT Edit Edit

Retrieve Save

Import Export

Tabbing
☒ No
☐ Yes
☐ Auto
☐ Tab At Start
☐ Tab By Distance

Tabbing Parameters

Quantity
NONE

Length
NONE

Height
NONE

Distance
NONE

Minimum Radius
0.00000

☒ Acc-n-Dec
☐ Metric
☐ Inline

☒ Plane Detect

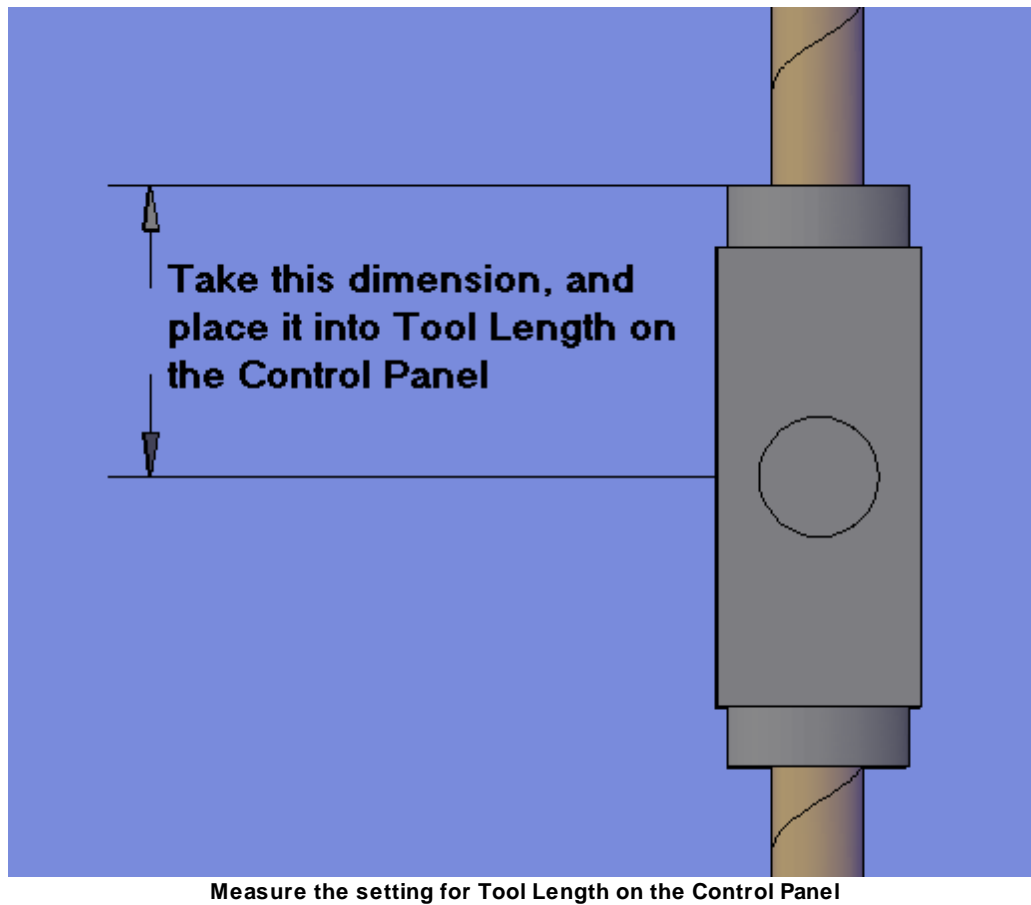
DOIT File
dotinfo.dat

OK Close

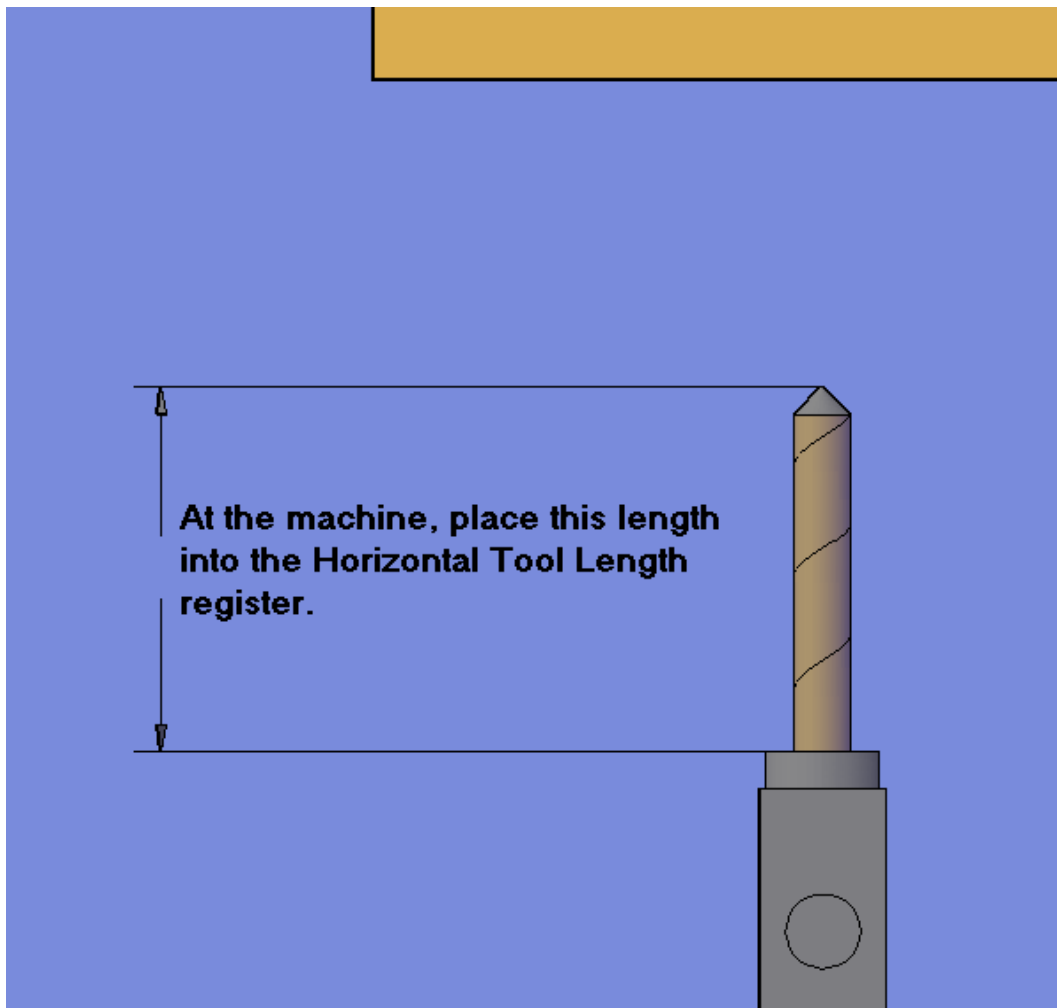
Geoshape Group Start Point Cut Sequence Seq. Undo Edit Code Mod Cycle Mod Tool NCVars

Combination of Spindle and Collet offsets

The Tool Length is set to the dimension of the center of the spindle to the face of the collet, which really only needs to be taken once. Set Plane Detect on. This is important.



Then make the cut, with the same settings as a Spindle offset tool path. The cut will be placed out away from the shape by the amount in the Tool Length parameter, and then at the machine, the operator must place the length of the tool in the Horizontal Tool Length offset register.



Place the Tool Length into the Horizontal Tool Length offset

When the machine sees the Horizontal Offset and the plane command (G18/G19) it will offset the tool path further by the length of the tool and place the tip of the cutter in the proper position.

This method is easier for both the programmer and operator from the standpoint that the programmer needs only one setting (Spindle to Collet distance) which will always stay the same, and for the operator that sets the tool offsets, the tool can be measured as it would normally be and that amount placed in an offset register just like a vertical tool.

Cutter Compensation

Cutter Compensation

☐ Yes ☐ No ☒ Both

Cutter Compensation will accept 1 of 3 selections: Yes, No, or Both, explained below.

Note: A thorough understanding of Cutter Compensation is very important. Consult your CNC control's Operator's Guide for more information on G41, G42, & G43 or your machines specific offset calls.

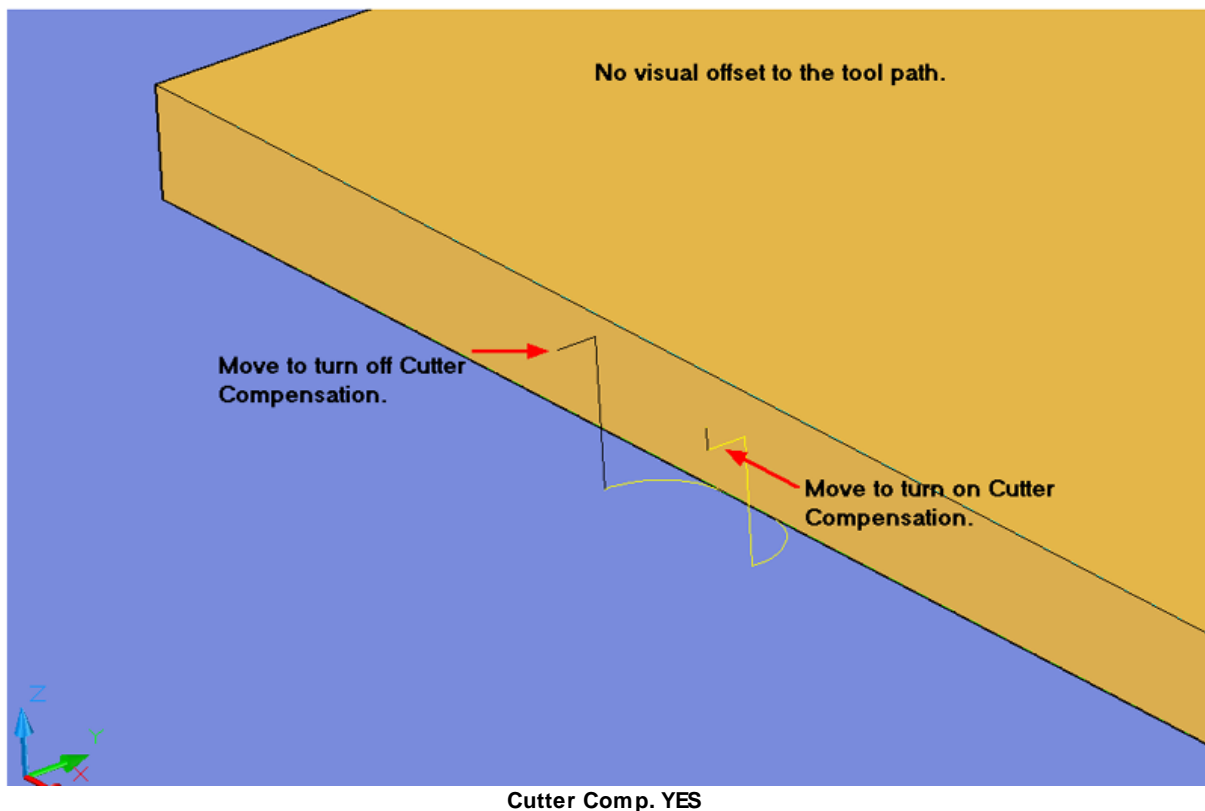
Note: CNC controls may require different code other than the ones listed. These will be determined by the post processor created for your specific CNC control type. Please refer to the post processor application notes that were included when the post processor was installed.

Cutter Comp YES

When using this option you will see the tool path directly on top of your part geometry.

The code that is produced reflects the part profile with compensation. At the machine, the entire tool radius must be placed into the offset register.

Your code will have the G41 or G42 with an offset register number, such as D01, and a move to turn off the compensation with the G40 code.

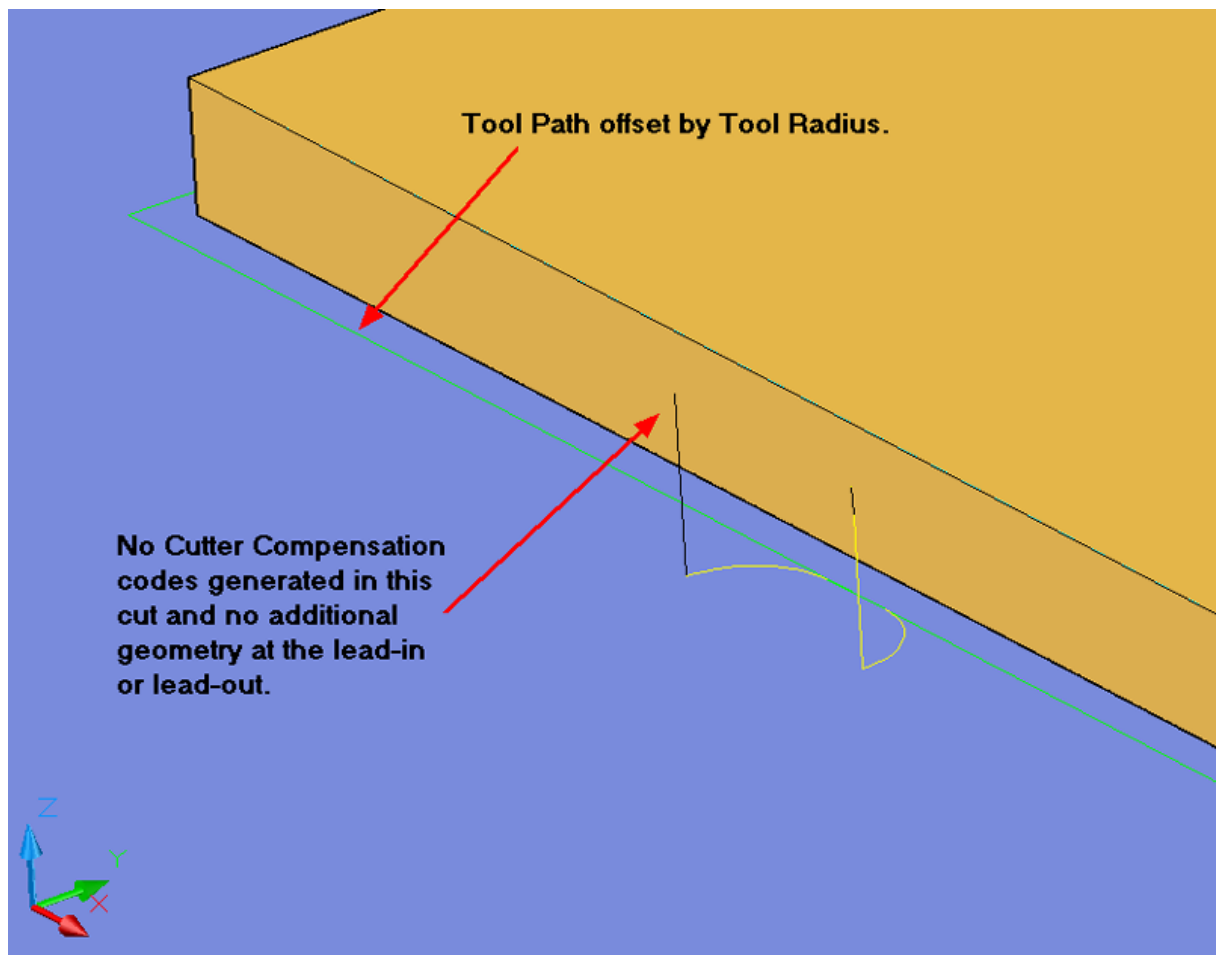


Cutter Comp NO

Using this option will generate a tool path that is offset by the radius of the tool as it appears on the Control Panel in the Tool Radius parameter.

This option does not generate a Cutter Compensation code (no G41/G42/G40 or D numbers).

Typically **No** is used for drilling, surface cutting, pocketing and engraving.



Cutter Comp. NO

Cutter Comp BOTH

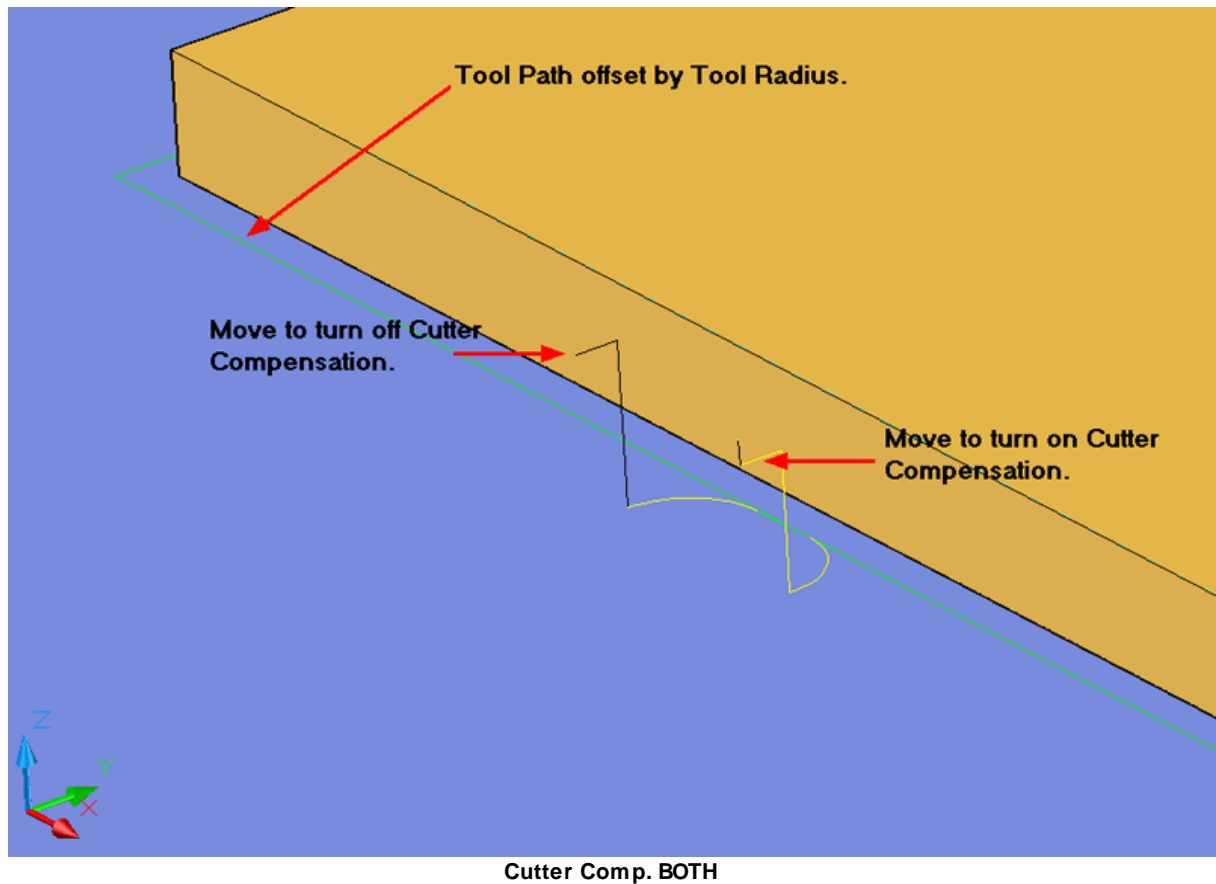
This option will show the tool path offset by the radius of the tool as it appears on the Control Panel in the Tool Radius parameter.

Note: Some CNC controls will not allow this option to be used. Please refer to the post processor application notes that were included when the post processor was installed to see the available options for your CNC control.

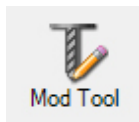
Both will generate a G41/G42 also, along with a D number and a G40 to cancel the compensation. The wear amount of the tool is then input into the offset register (D#) at the machine.

This option provides the best results because you can visually inspect the offset on the screen, and compensate for tool wear. Typically used for all contour cutting.

The B for Both choice is useful, because you input a D value at the machine that is the difference between what Router-CIM offsets and what the tool radius actually is (usually a very small amount). Measure the tool radius and add or subtract that value from what was offset (nominal radius), and then place that difference into the appropriate D register (the offset page at the controller). Only by using Cutter Compensation can you adjust for sharpened or worn bits as well as to parts cut with different sized bits.



Modifying a Tool



Modifying Tools is a common and necessary practice in Router-CIM. You can make these changes temporarily, in a current drawing, or permanently, in your machine specific default drawing. You can even create and use a new tool list if desired.

There will be times when just changing the diameter and radius are enough to get the job done. However if you find that you are making changes often, making a new tool may be a better solution.

There are three sections to this chapter.

- [Temporary Editing](#)
- [Permanent Editing](#)
- [Moving an Item in the List](#)

Temporary Editing

When you start Router-CIM, a blank drawing is inserted into your current drawing session. That blank drawing contains all of the blocks and lists that Router-CIM needs in order to display the tool, cycle and status information on the Control Panel. Once the drawing is saved with that information in it, any temporary edits you made to Router-CIM are available in that drawing for future use. Router-CIM retains in the cut knowledge, any edits you made in order to create a tool path for the cutting condition you needed.

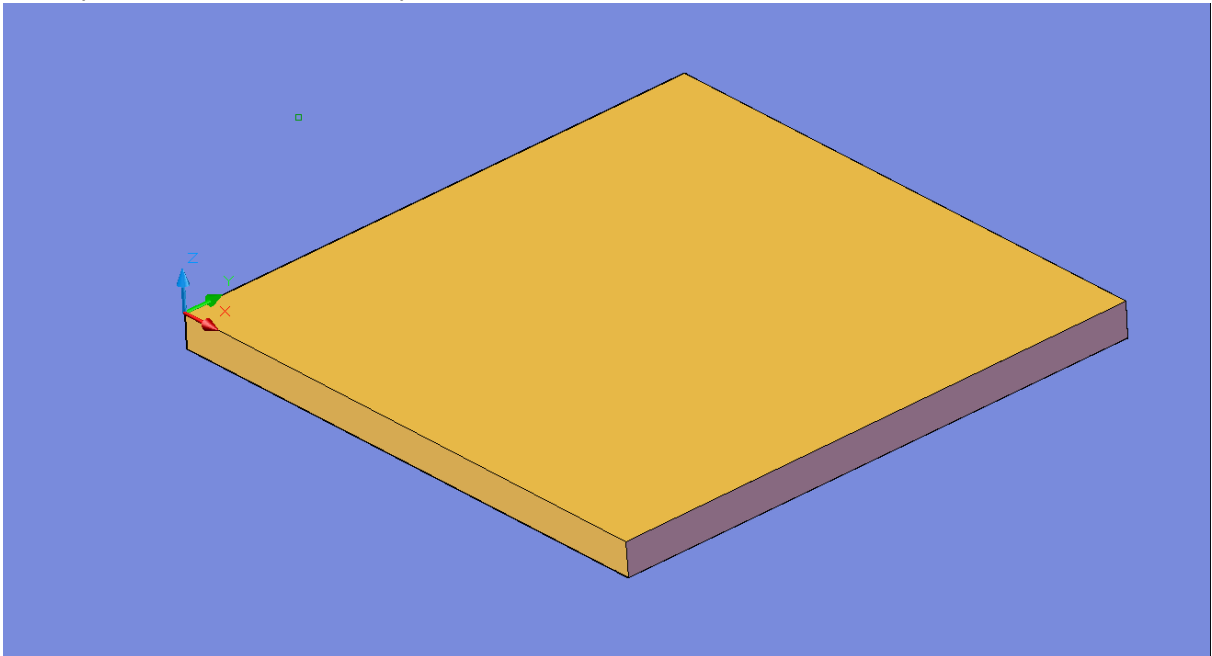
If you modify a tools' parameters, then make a cut, that cut contains all the modified parameters for that tool.

The key here is that this information is only available in the drawing if the tool paths are saved with the modified data in them, and only that particular drawing has the changed parameters.

This might be enough if you are just modifying a tool number, offset, comment or temporarily changing a tool size.

To demonstrate the changes that are stored and how they are stored, the following example is shown.

A new part is created, and for this part we will need a new tool.



The default tool show in Router-CIM in our current session is a .5" Router-BIT. This is the tool we are going to temporarily modify into a 1.5" diameter shaper tool.

Default Tool Settings

© 2025 CIM-TECH.COM, Inc.

CIM-Tech Router-CIM Control Panel

Description

Tool Information
ROUTER-BIT_500

Tools

Tool Number / Comment
"4"
"ROUTER-BIT 1.5 DIA. SHAPER"

CRC Offset
autocr

Spindle Direction
CW

Tool Diameter
1.50000

Tool Radius
0.25000

Tool Length
3.00000

4 Axis Safe

Tool Type

Category

Vertical Offset

Horizontal Offset

Cutter Bridge

Aggregate Offset
☒ Spindle ☐ Collet

Cutter Compensation
☐ Yes ☐ No ☒ Both

Cycle Information

Offset Dim
firstxy xycutloc

Cut Side
outside

Cut Direction
CW

Round Corners
n

Lead In
MULTILI

Lead Out
MULTILO

Lead Size
leadscl

Lead Feed

Overlap Amount
AUTO

XY Stock Allowance

Z Stock Allowance

Corner Punch
N

Status Information

Safety Plane
0.25000

Depth Per Pass
0.75000

Total Cut Depth
-0.75000

Feedrate/Spindle Speed

Feedrate
300.00000

Spindle Speed
10000.00000

Surface FPM
NONE

Units Per Revolution
NONE

Calculate

Before Codes

After Codes

Oscillation Amount
0.00000

Sort By Rank #

PLUNGE OUTSIDE

Reset Cycle Settings to Default

Knowledge / Settings

Knowledge
Select Knowledge
CURRENT

DOIT Edit Edit

Retrieve Save

Import Export

Tabbing
☒ No
☐ Yes
☐ Auto
☐ Tab At Start
☐ Tab By Distance

Tabbing Parameters

Quantity
NONE

Length
NONE

Height
NONE

Distance
NONE

Minimum Radius
0.00000

☒ Acc-n-Dec
☐ Metric
☐ Inline
☐ Plane Detect

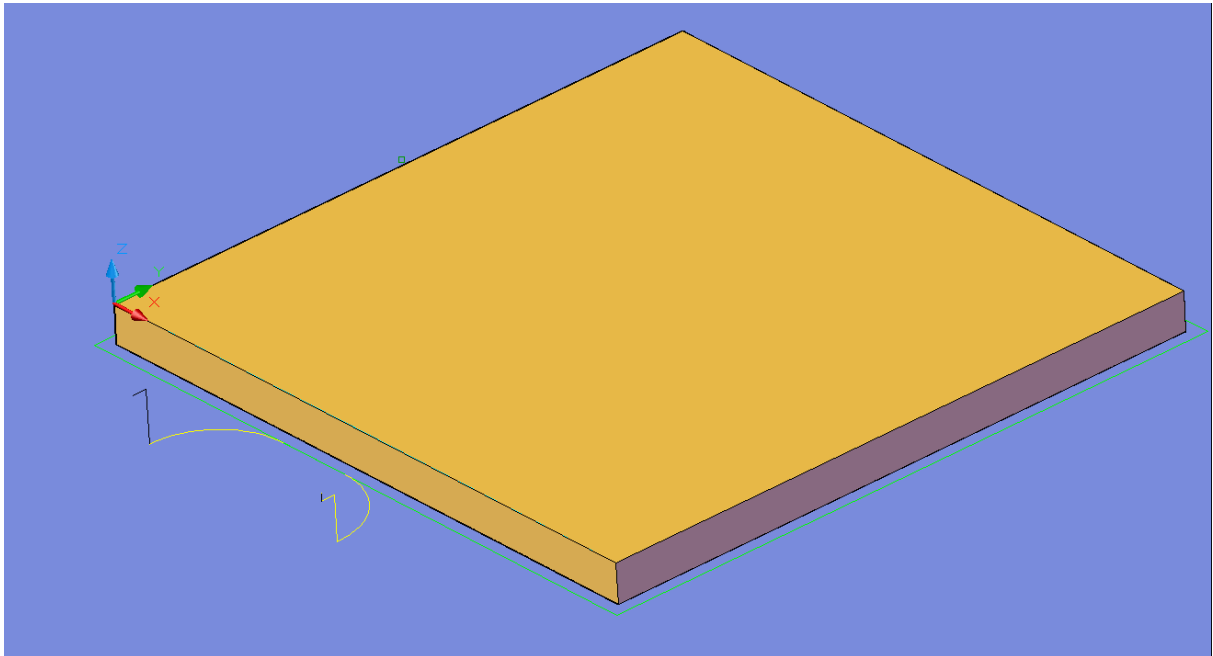
DOIT File
dotinfo.dat

Geoshape Group Start Point Cut Sequence Seq. Undo Edit Code Mod Cycle Mod Tool NCVars

OK Close

Temporary Edit to Tool

Once those changes are made, we can set the Feedrate, Spindle Speed, Safety Plane and any other cycle or status parameters and then make the cut.



New, temporary, parameters applied to tool path

When the drawing is saved, with the tool path, all the parameters that were changed will be available again by retrieving the knowledge from the cut.

If a New drawing is opened, however, none of the changes that were made will be available because they were only edited in the control panel and only saved within the drawing where they were used.

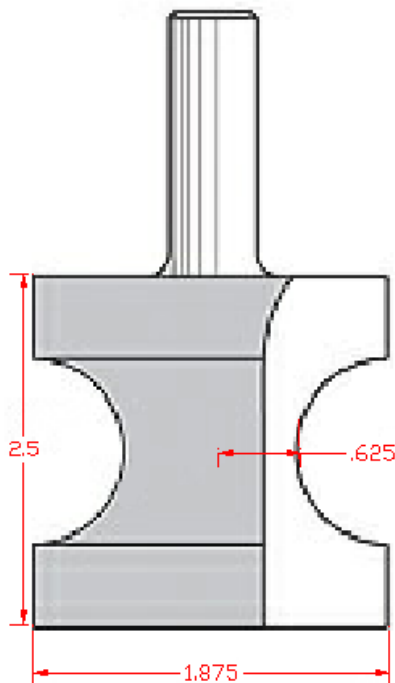
If the tool you modified is going to be used over again at a later time, a more permanent solution is needed to avoid having to enter the tool parameters over and over again. Saving these parameters requires Permanent Editing.

Permanent Editing

Making permanent edits to the tool list involves making the changes to the tool in the permanent editing section, and then adding the tool to the list and filing the list so that it is available again in other drawings.

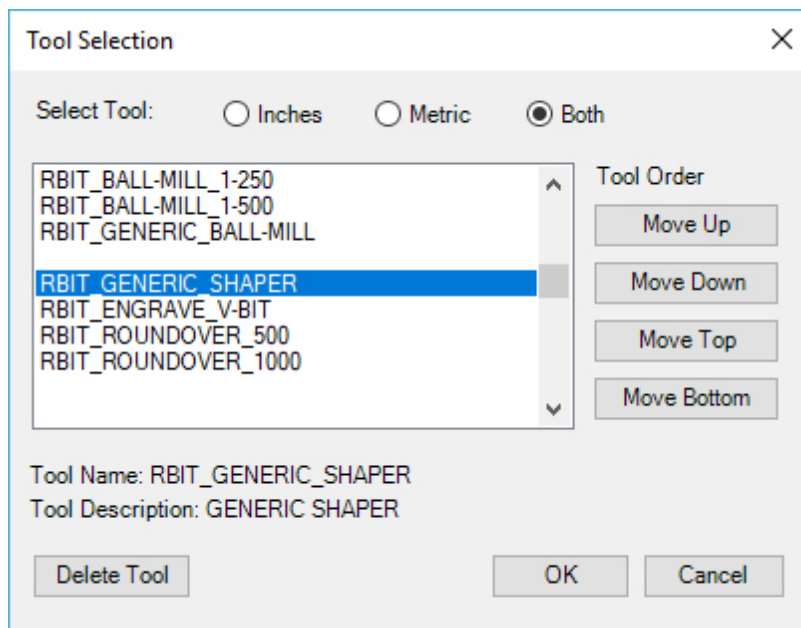
In order to show Permanent Editing, we will create a new tool in the tool list, then file the list and open a new Router-CIM session to show the tool available. You should read through all the steps involved before trying this and then come back and step through them one at a time until your results are the same.

The new tool created in this example is going to be a 1.875" Diameter shaper type tool that looks like the following illustration.



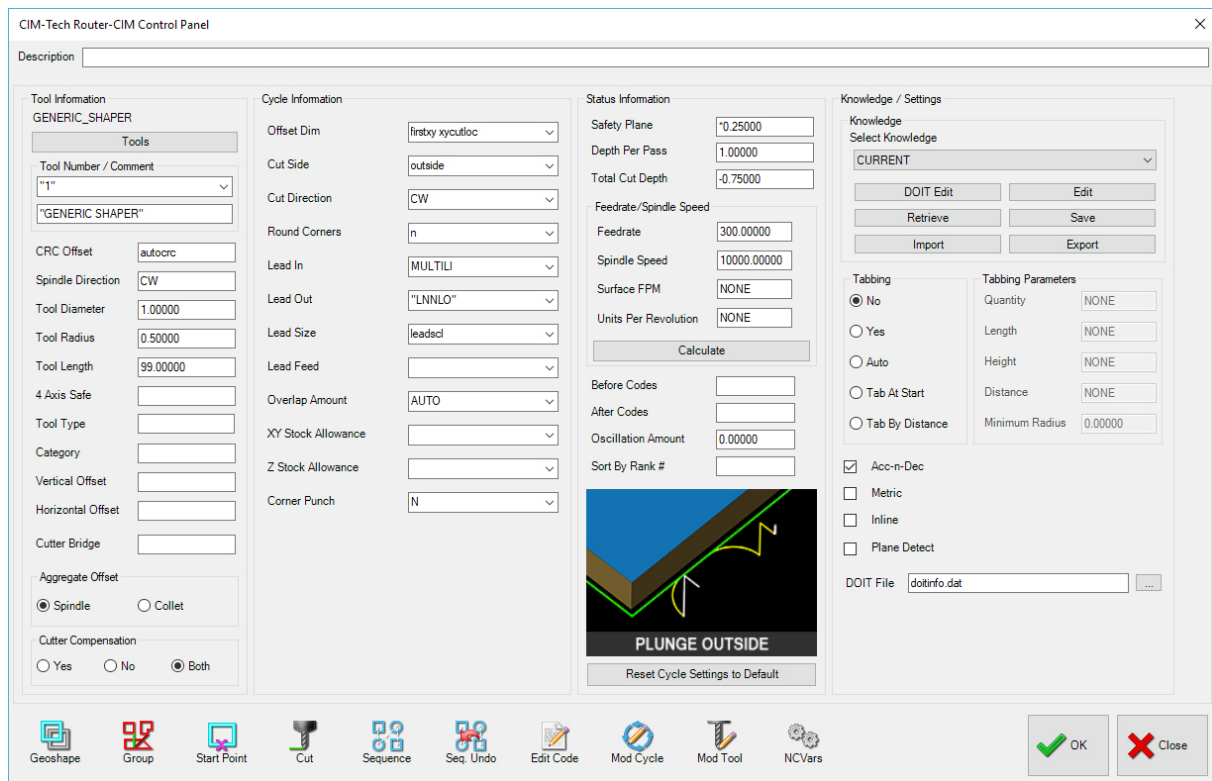
In order to make this tool, a new, blank, drawing is opened up and Router-CIM is loaded. Then we start with the Control Panel and pick a tool from the tool list that is somewhat close to what we want.

Select the TOOLS button and in this case the RBIT_GENERIC_SHAPER is selected.



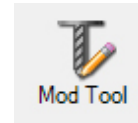
Select a Generic Shaper tool.

Once that tool is selected, click on OK to move back to the Control Panel with that tool as the current tool.



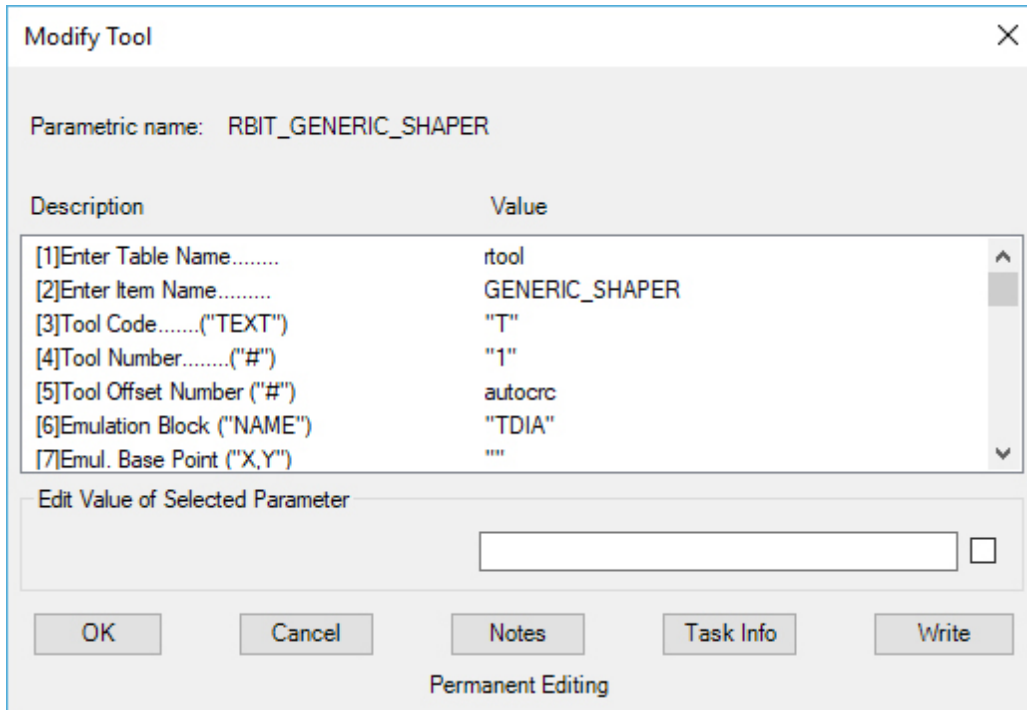
The Generic Shaper is now the current tool

Next we will modify the tool parameters to create the new tool for the list.



Select Mod Tool in the lower right portion of the Control Panel:

Mod Tool displays all the tool parameters, some of which will be changed to create the new tool.



Description	Value
[1]Enter Table Name.....	rtool
[2]Enter Item Name.....	GENERIC_SHAPER
[3]Tool Code.....("TEXT")	"T"
[4]Tool Number.....("#")	"1"
[5]Tool Offset Number ("#")	autocrc
[6]Emulation Block ("NAME")	"TDIA"
[7]Emul. Base Point ("X,Y")	""

Edit Value of Selected Parameter

☐

OK Cancel Notes Task Info Write

Permanent Editing

The first item to change is the Item Name (#2 in the list). Select this item with your mouse and then at the bottom of the window you will see the value to be changed.

Description	Value
[1]Enter Table Name.....	rtool
[2]Enter Item Name.....	GENERIC_SHAPER
[3]Tool Code.....("TEXT")	"T"
[4]Tool Number.....("#")	"1"
[5]Tool Offset Number ("#")	autocrc
[6]Emulation Block ("NAME")	"TDIA"
[7]Emul. Base Point ("X,Y")	""

Edit Value of Selected Parameter

[2]Enter Item Name.....

OK Cancel Notes Task Info Write

Permanent Editing

Turn on the Caps Lock and type in SHAPER_TOOL_1875 and then Press ENTER on the keyboard.

Description	Value
[1]Enter Table Name.....	rtool
[2]Enter Item Name.....	GENERIC_SHAPER
[3]Tool Code.....("TEXT")	"T"
[4]Tool Number.....("#")	"1"
[5]Tool Offset Number ("#")	autocrc
[6]Emulation Block ("NAME")	"TDIA"
[7]Emul. Base Point ("X,Y")	""

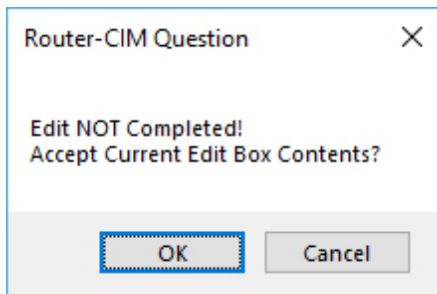
Edit Value of Selected Parameter

[2]Enter Item Name.....

OK Cancel Notes Task Info Write

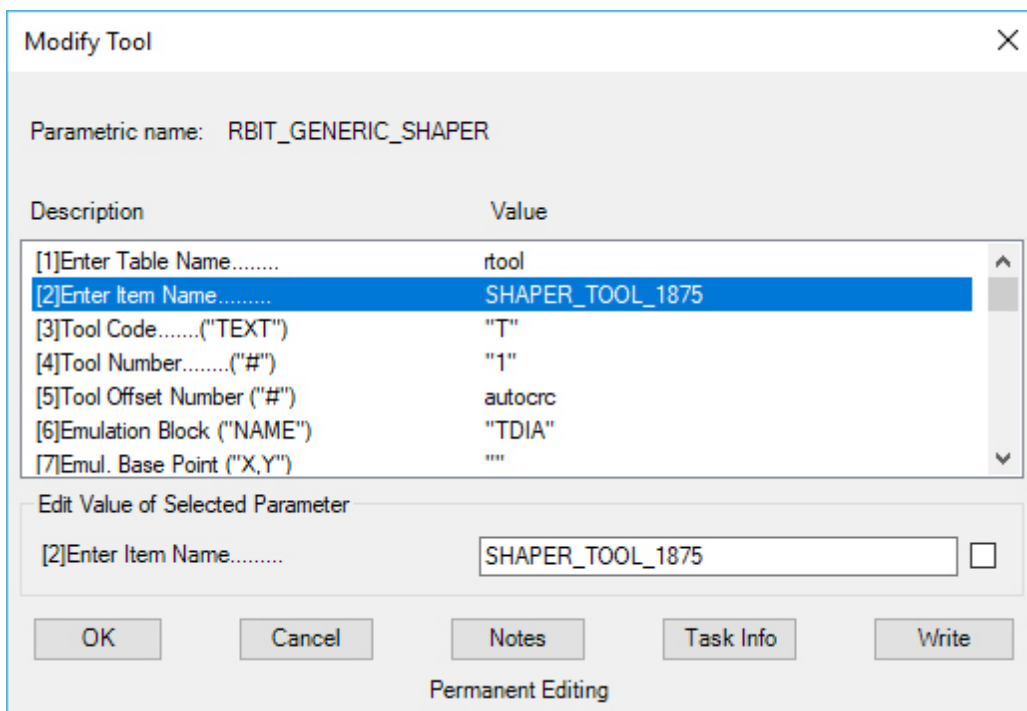
Permanent Editing

If you do not press ENTER, you will see the following notice:



Select YES if you see this notice.

The parameter window should then show the changed value.



Next we will select the Tool Number (#4 in the list) and change the number in quotes from "1" to "4". Again, select the item with your mouse and it will be available below to edit.

Modify Tool

Parametric name: RBIT_GENERIC_SHAPER

Description	Value
[1]Enter Table Name.....	rtool
[2]Enter Item Name.....	SHAPER_TOOL_1875
[3]Tool Code.....("TEXT")	"T"
[4]Tool Number.....("#")	"1"
[5]Tool Offset Number ("#")	autocrc
[6]Emulation Block ("NAME")	"TDIA"
[7]Emul. Base Point ("X,Y")	""

Edit Value of Selected Parameter

[4]Tool Number.....("#") ☐

OK Cancel Notes Task Info Write

Permanent Editing

Change the number inside the quotes from "1" to "4" and then press ENTER and you will see the change in the list.

Modify Tool

Parametric name: RBIT_GENERIC_SHAPER

Description	Value
[1]Enter Table Name.....	rtool
[2]Enter Item Name.....	SHAPER_TOOL_1875
[3]Tool Code.....("TEXT")	"T"
[4]Tool Number.....("#")	"4"
[5]Tool Offset Number ("#")	autocrc
[6]Emulation Block ("NAME")	"TDIA"
[7]Emul. Base Point ("X,Y")	""

Edit Value of Selected Parameter

[4]Tool Number.....("#") ☐

OK Cancel Notes Task Info Write

Permanent Editing

The next item we need to change in this case is the Describe Tool (#20 in the list).

Modify Tool

Parametric name: RBIT_GENERIC_SHAPER

Description	Value
[20]Describe Tool...("TEXT")	"GENERIC SHAPER"
[21]Tool Diameter *TW* (#)	1.00000
[22]Tool Length *TL* (#)	99.00000
[23]Tool Radius *TR* (#)	0.50000
[24]Tool Cut Depth *TD* (#)	1.00000
[25].....(reserved)	
[26].....(reserved)	

Edit Value of Selected Parameter

[20]Describe Tool...("TEXT")

OK Cancel Notes Task Info Write

Permanent Editing

Select that and change it from "GENERIC SHAPER" to "SHAPER TOOL 1.875 DIA.", then press ENTER to see the change in the list.

Modify Tool

Parametric name: RBIT_GENERIC_SHAPER

Description	Value
[20]Describe Tool...("TEXT")	"SHAPER TOOL 1.875 DIA."
[21]Tool Diameter *TW* (#)	1.00000
[22]Tool Length *TL* (#)	99.00000
[23]Tool Radius *TR* (#)	0.50000
[24]Tool Cut Depth *TD* (#)	1.00000
[25].....(reserved)	
[26].....(reserved)	

Edit Value of Selected Parameter

[20]Describe Tool...("TEXT")

OK Cancel Notes Task Info Write

Permanent Editing

We will then change:

- Tool Diameter (#21 in the list) to 1.875.
- Tool Length (#22 in the list) to 2.5
- Tool Radius (#23 in the list) to .625

- Tool Cut Depth (#24 in the list) to 2.5. This relates to the 'Depth Per Pass' field.

The list should then look like this:

Description	Value
[20]Describe Tool...('TEXT')	"SHAPER TOOL 1.875 DIA."
[21]Tool Diameter *TW* (#)	1.875
[22]Tool Length *TL* (#)	2.5
[23]Tool Radius *TR* (#)	0.625
[24]Tool Cut Depth *TD* (#)	2.5
[25].....(reserved)	
[26].....(reserved)	

Edit Value of Selected Parameter

[24]Tool Cut Depth *TD* (#)

OK Cancel Notes Task Info Write

Permanent Editing

Once the changes in this editor are made, select OK to add a new tool to the list. The following window appears.

Rename/Add Item to List

List Name:

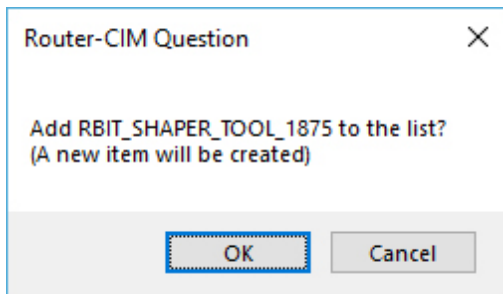
- RBIT_RTOOL_MFENDML
- RBIT_ROUTER-BIT_0625
- RBIT_ROUTER-BIT_100
- RBIT_ROUTER-BIT_125
- RBIT_ROUTER-BIT_1875
- RBIT_ROUTER-BIT_250
- RBIT_ROUTER-BIT_3125
- RBIT_ROUTER-BIT_375
- RBIT_ROUTER-BIT_4375
- RBIT_ROUTER-BIT_500

Current Name: RBIT_GENERIC_SHAPER

New Name:

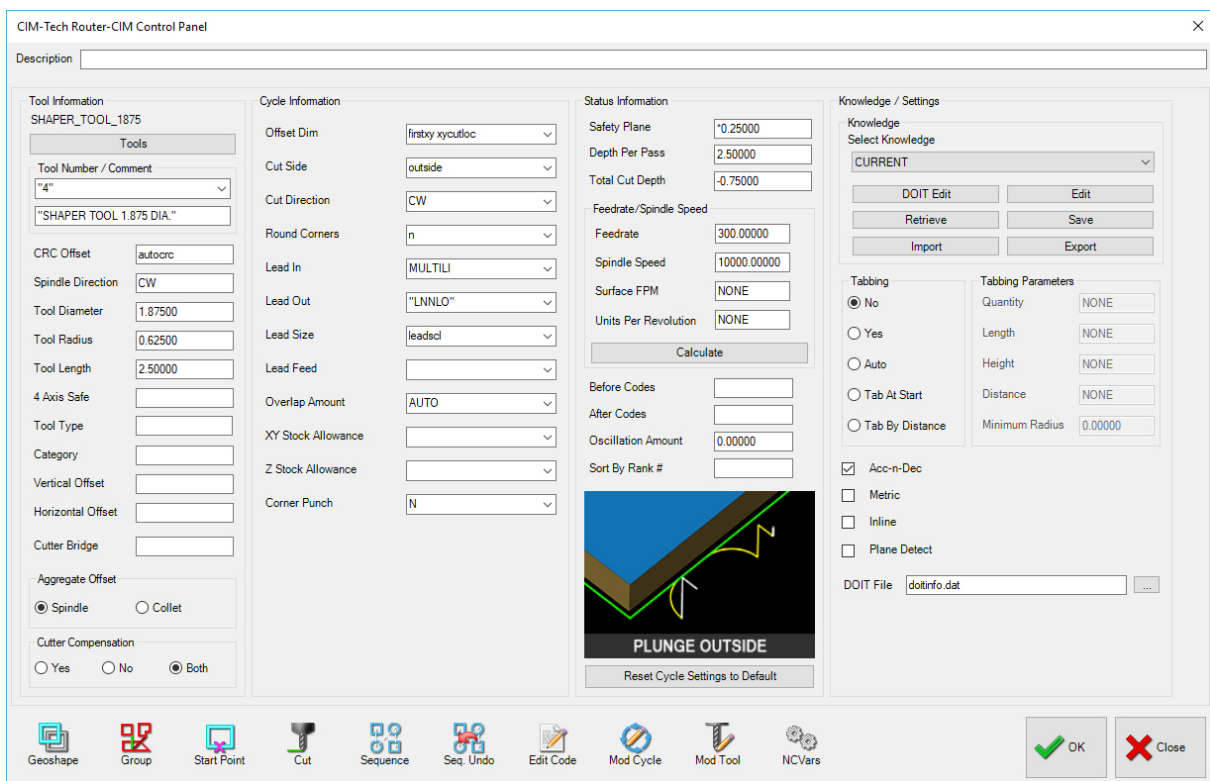
OK Cancel

This shows you that you are going to either Rename or Add a new item to the list. In this case it will be a new item.
Select OK.

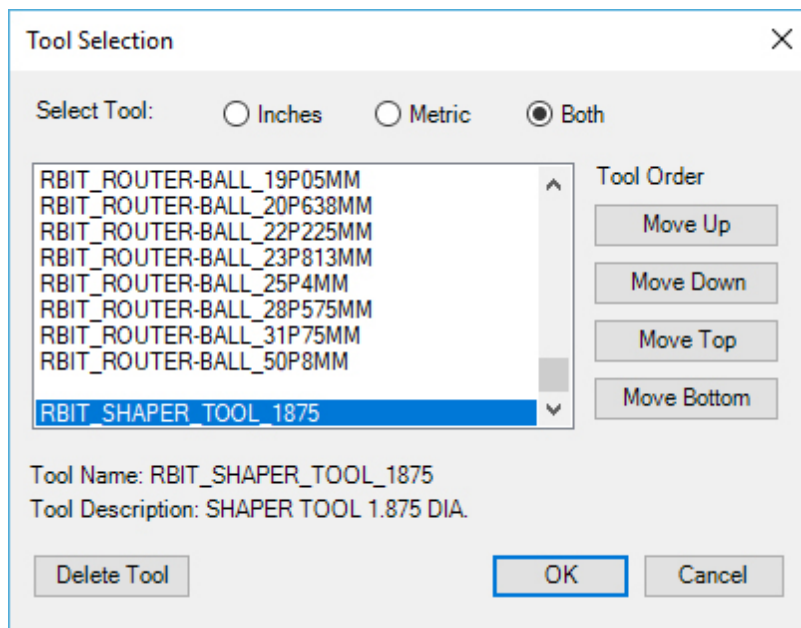


You get a final window asking if you want to add a new item to the list. Select 'YES'.

Then you will see the Control Panel with the new tool displayed.



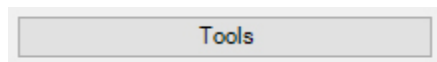
At this point the tool is added to the list, and the list is filed to the hard drive. If you select the tool list now, you can see the new tool at the very bottom of the list.



In the next section we will move that tool up in the list so that it is easier to find.

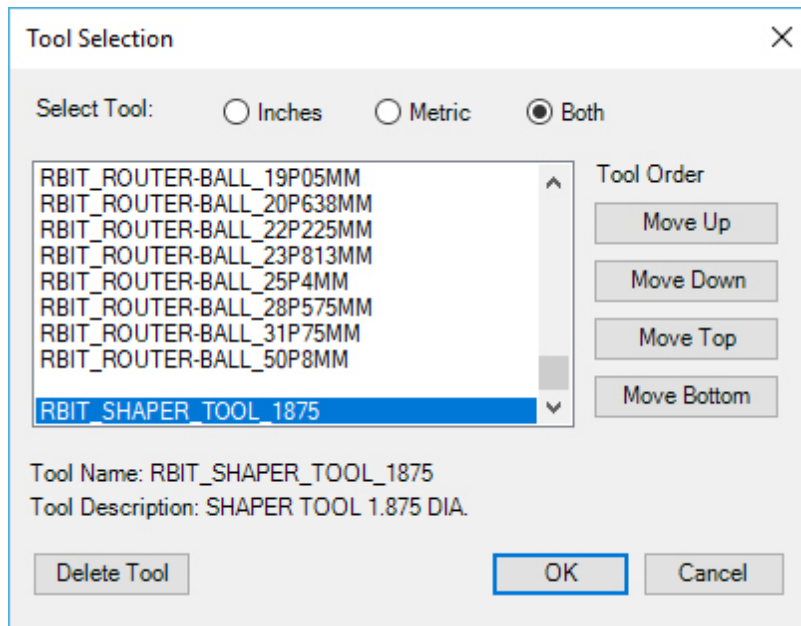
Moving an Item in the List

To move tools to new locations or organize tools in the Tool List open the tool list by selecting the 'Tools' button in the control panel:



The lists can be re-arranged and annotated so that the list has a sense of grouping items together and commenting as to what the items are in the list.

When the tool list opens, highlight the tool you would like to move:



On the right side of the 'Tool Selection' dialog box, you can use the buttons to move an item in the list:

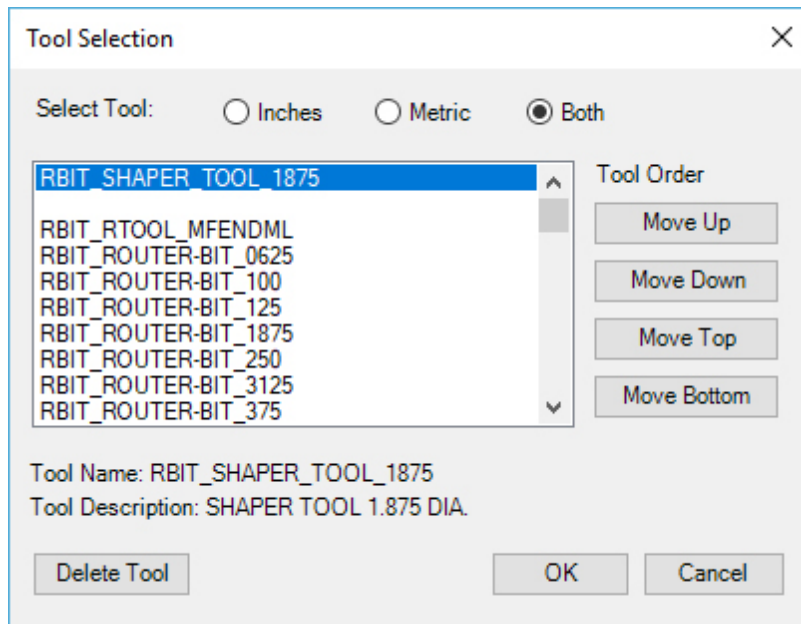
Move Up: Moves the selected tool up 1 position

Move Down: Moves the selected tool down 1 position

Move Top: Moves the selected tool to the top of the list

Move Bottom: Moves the selected tool to the bottom of the list

Once you have placed the tool in the desired location, select 'OK' and this will save the tool in the list:



Delete Tool

If there are tools that are no longer used, you can delete tools from the list by selecting the tool you no longer need and then selecting the 'Delete Tool' button at the lower left of the 'Tool Selection' dialog box. Select 'OK' when finished to save the changes or 'Cancel' to not save the changes.

Changing Tool Parameters

When changing the tool parameters, it is important to know the definition of the parameters you need to change to get the desired results. Not all of these parameters will have to be changed with every tool, but you should be familiar with the settings and their definitions.

Description	Value
[1]Enter Table Name.....	rtool
[2]Enter Item Name.....	GENERIC_SHAPER
[3]Tool Code.....("TEXT")	"T"
[4]Tool Number.....("#")	"1"
[5]Tool Offset Number ("#")	autocrc
[6]Emulation Block ("NAME")	"TDIA"
[7]Emul. Base Point ("X,Y")	""

Edit Value of Selected Parameter

OK Cancel Notes Task Info Write

Permanent Editing

(01) ENTER TABLE NAME... This is the table name of the tool list and normally is never changed.

(02) ENTER ITEM NAME... This is the name of the tool as it will appear on the tool list. You should try to keep the name format of similar tools similar to each other, so they are easy to find in the list (refer to current list for reference). You cannot use spaces, or special characters in this version of the tool name. Only Underscore and Hyphen are allowed.

(03) TOOL CODE... This is the code put out for individual tools, usually the tool definitions are accounted for in the post processor so changes are not necessary. Unless you are directed to change this parameter in your application notes, you should not change this parameter at all.

(04) TOOL NUMBER... This is the default number of the tool as it appears in the Control Panel. If you have a tool that is always the same number, place the number here so that it is the default. The number must be set inside quotation marks in order to generate the correct code.

(05) TOOL OFFSET NUMBER... This is the same as the offset number on the Control Panel. Normally you will not need to change the field from 'autocr' unless you wish to have a specific offset number for a particular tool. This will output as the "D" value with the number you specify in this field.

(06) - (17) EMULATION BLOCK NAME, AND EMULATION XY SCALES... These values control the block to insert and the size of the Tool Check tool. You can show a detailed Tool Check by drawing a 3D tool and storing it as a Block in your default or current drawing. Put the block name in the Emulation Block Name field. During the Tool Check command, that block will be inserted. A task called Tdsize will use the tool's diameter for the emulation X and Y scale factors. These are for advanced use of the Tool Check function and are not normally used for regular programming.

(18) TOOL ROTATION... This is the same field that appears in the Control Panel as Spindle Dir. If you have counter-clockwise spindles, you should change this field when you are using CCW bits in your spindle. This field generates an M03 or M04 code for clockwise and counter-clockwise normally.

(19) TOOL TIP DISTANCE TO ZERO... This is the distance that you need to compensate for if you have a tool whose tip is not flat. If you touch off the tip of the tool to zero, but there is an angled or rounded surface to the end of the tool, such as a drill bit or a ball mill, you can input the distance to the desired cutting surface here.

(20) DESCRIBE TOOL... This will be the description of the current tool as it will appear in the NC code file (of your router supports comments). This field can hold any description or character, however it must be enclosed in quotation marks.

(21) TOOL DIAMETER... This is the field that appears in the Control Panel as Tool Diam. The diameter of the tool input here should be the outside diameter of the carbide.

(22) TOOL LENGTH... This is the same field that appears in the Control Panel, if you wish to set the default Tool Length input a number in this field.

(23) TOOL RADIUS... This is the same field that appears in the Control Panel as Tool Rad.

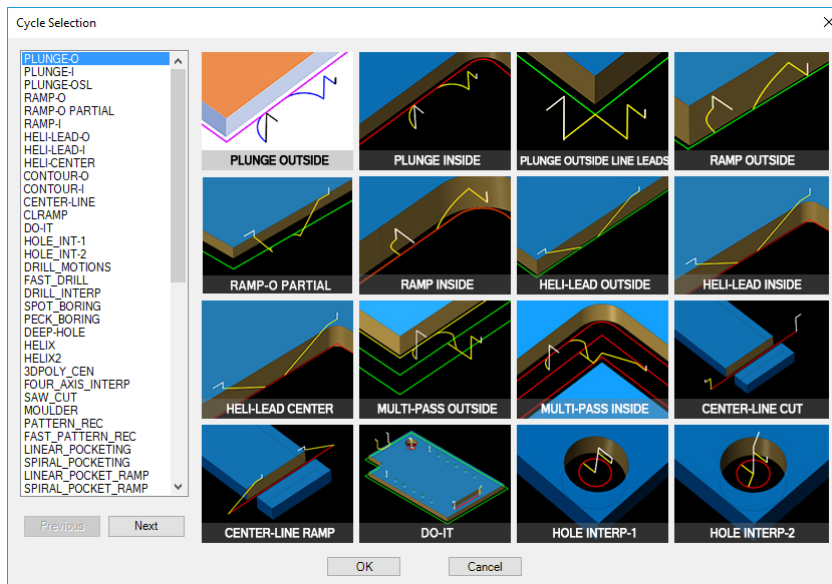
(24) TOOL CUT DEPTH... This is the same field that appears in the Control Panel as the Depth Per Pass field. This is the maximum depth you wish the tool to be able to cut in the Z direction in any single pass, thus extending tool life.

(25) - (26) These fields are reserved and should not be edited in normal use.

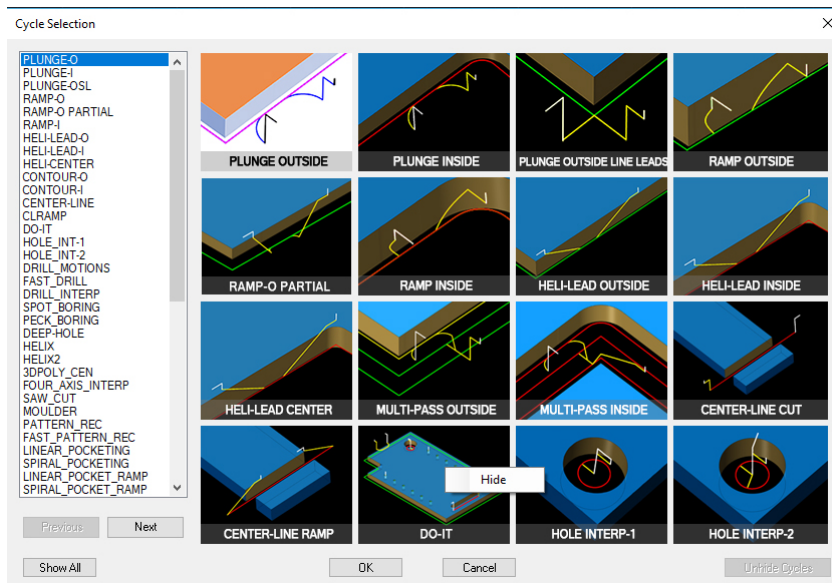
(27) This field should normally be set to "Y" and should not be edited in normal use.

(28) MACHINE CUTTER COMPENSATION... This is the field where you can specify the default value for the Cutter Comp. Selection in the Control Panel. Valid entries are "Y", "N", or "B" for Yes, No, or Both. Please refer to the section on Cutter Compensation for further details.

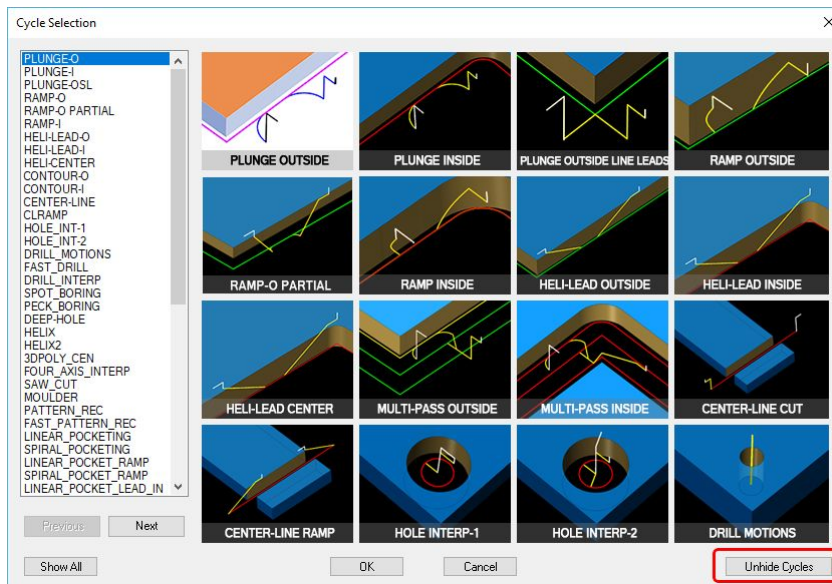
(29) FEED DISTANCE TO MATERIAL... This is the point at which your machine tool switches from rapid traverse to programmed feed rate. This field is set to .1 and should not be set any lower than this! The machine tool will rapid to this point and then the next line of code will have to use a feed rate code if it is going to enter the material.



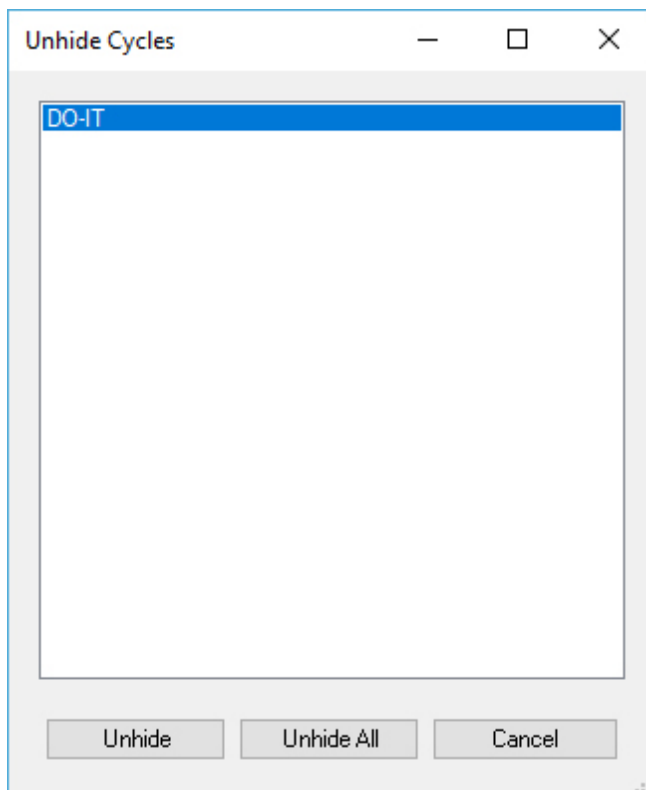
If you do not use or want to have a cycle visible, you can hide the cycle image. To hide the cycle image, right-click on the cycle you want and select the 'Hide' option:



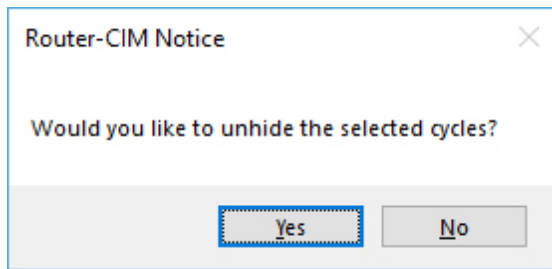
To bring the cycle back, in the lower right of the Cycle Selection screen, select the 'Unhide Cycle' button:



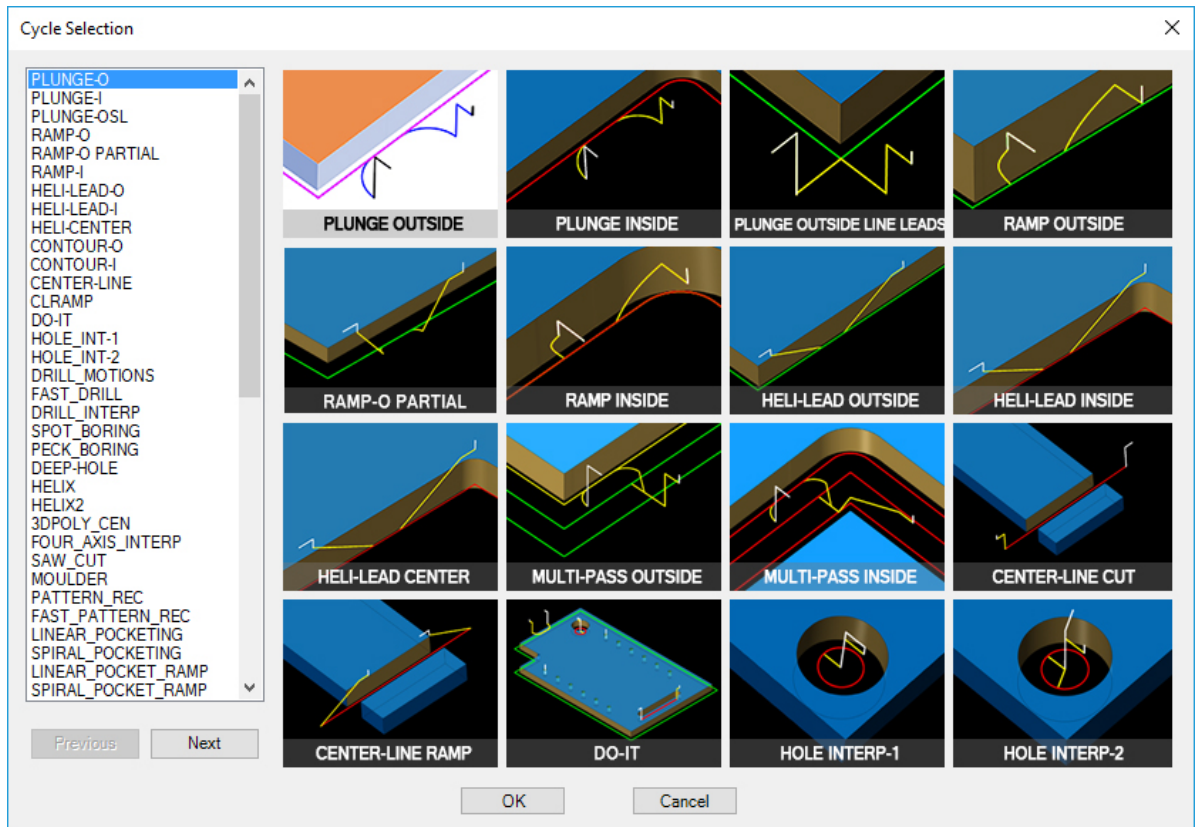
This will open the 'Unhide Cycles' screen. From here you can select one or multiple using your CTRL or SHIFT keys. If you want to unhide all the hidden cycles, select the 'Unhide All' button.



Once you select 'Unhide' or 'Unhide All' you will be asked to confirm. Select 'Yes' to unhide or 'No' to leave them hidden.

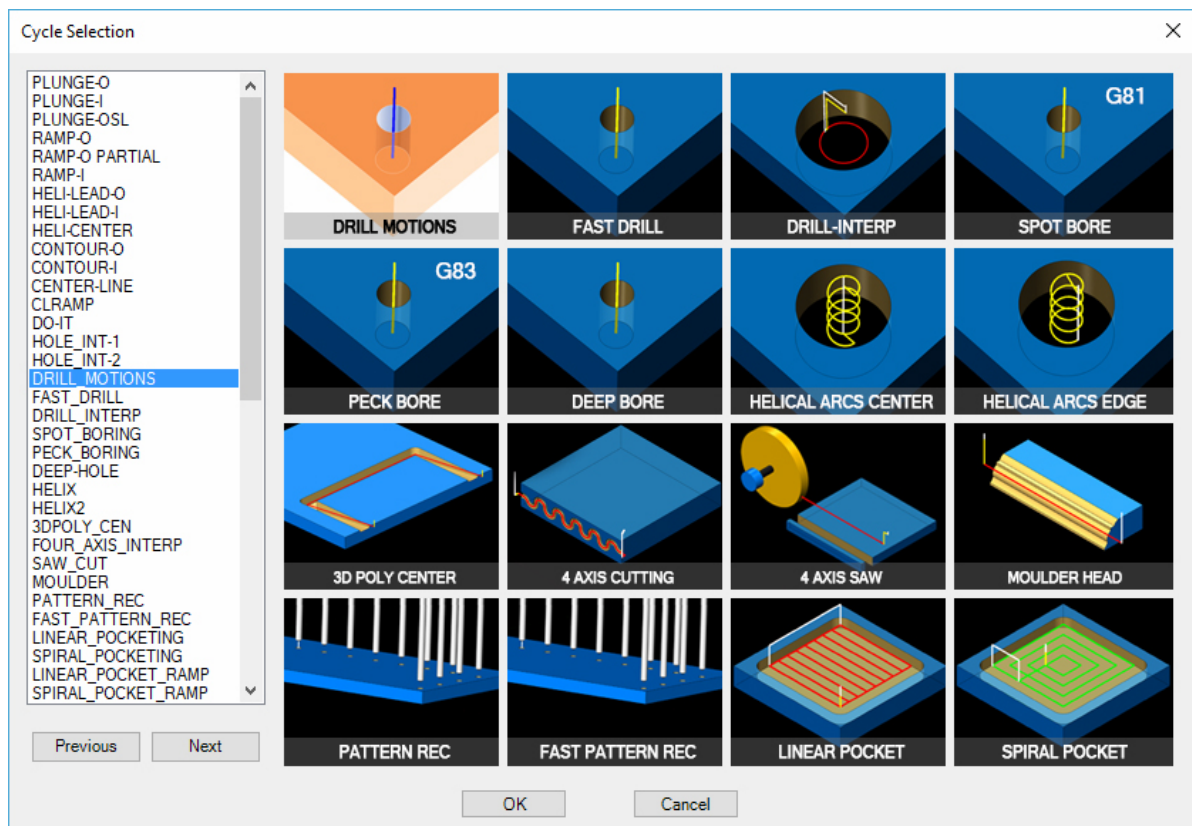


Each of the four Cycle screens can show up to 16 cycles. You may toggle back and forth between the four cycle screens with the Next and Previous buttons.



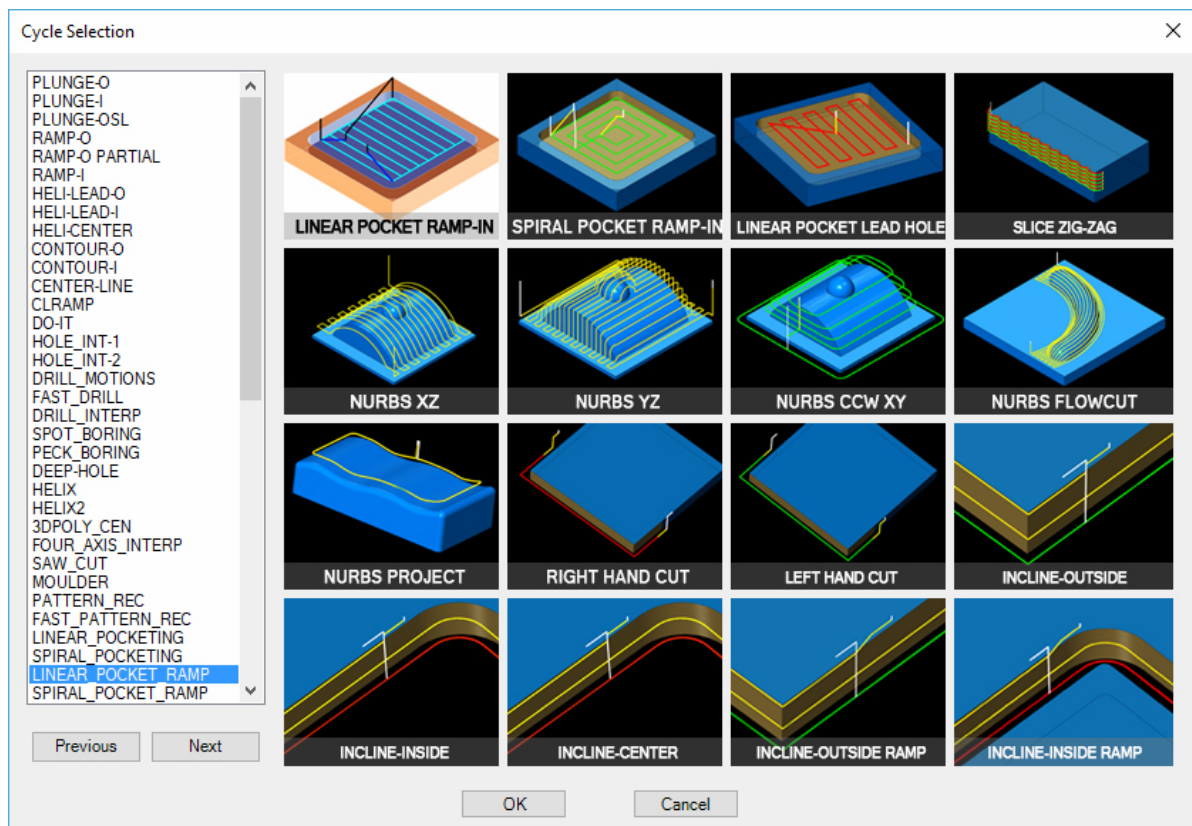
Cycle Menu 1

The second Cycle screen is shown below:



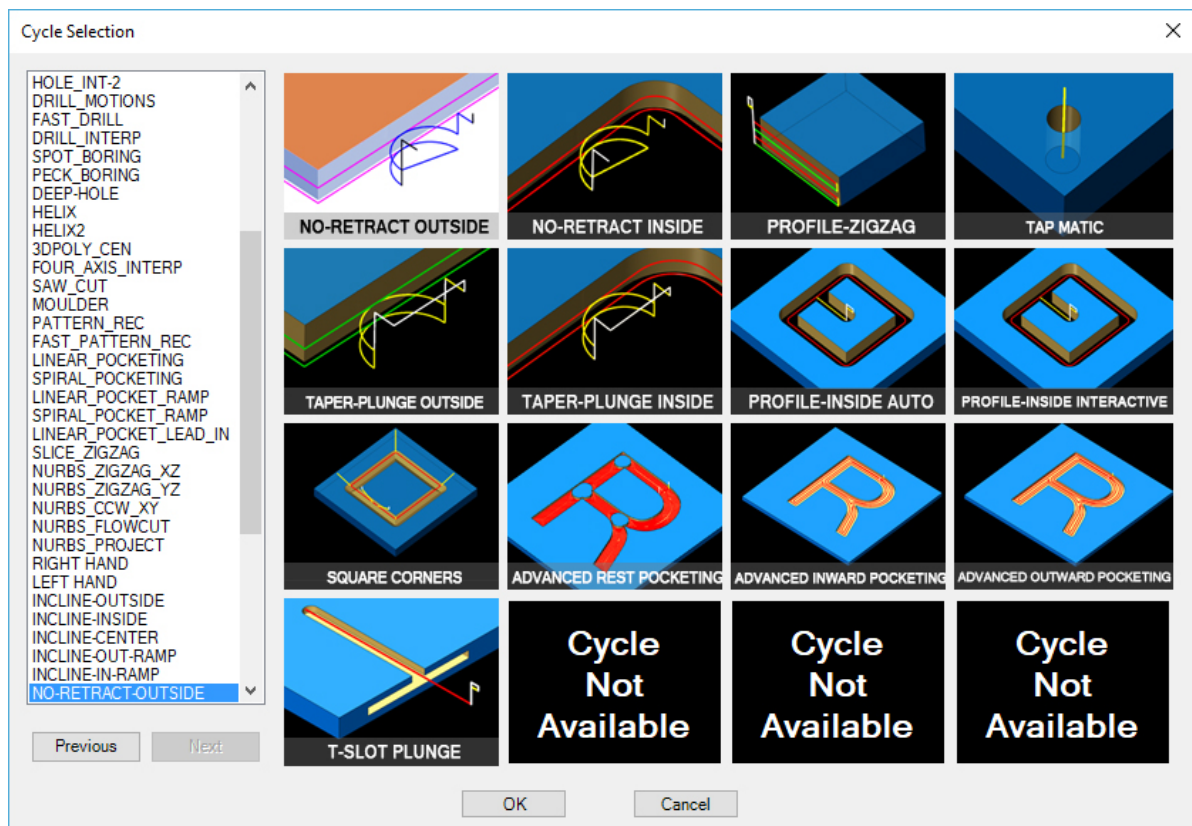
Cycle Menu 2

The third Cycle screen is shown below:



Cycle Menu 3

The fourth Cycle screen is shown below:



Cycle Menu 4

To understand the data that must be input to make each cycle work correctly, we need to analyze each cycle along with the Cycle Information column. The cycles may be broken into common groups which require similar inputs in the Cycle Information column.

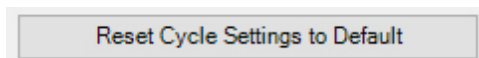
Cycle Descriptions

Each of the cut cycles in Router-CIM have particular settings to allow their use with minimal setup. Still, some explanation of these cycles is necessary to enable a full understanding of their parameters.

Each of the parameters available for each cycle is listed and the possible entries shown.

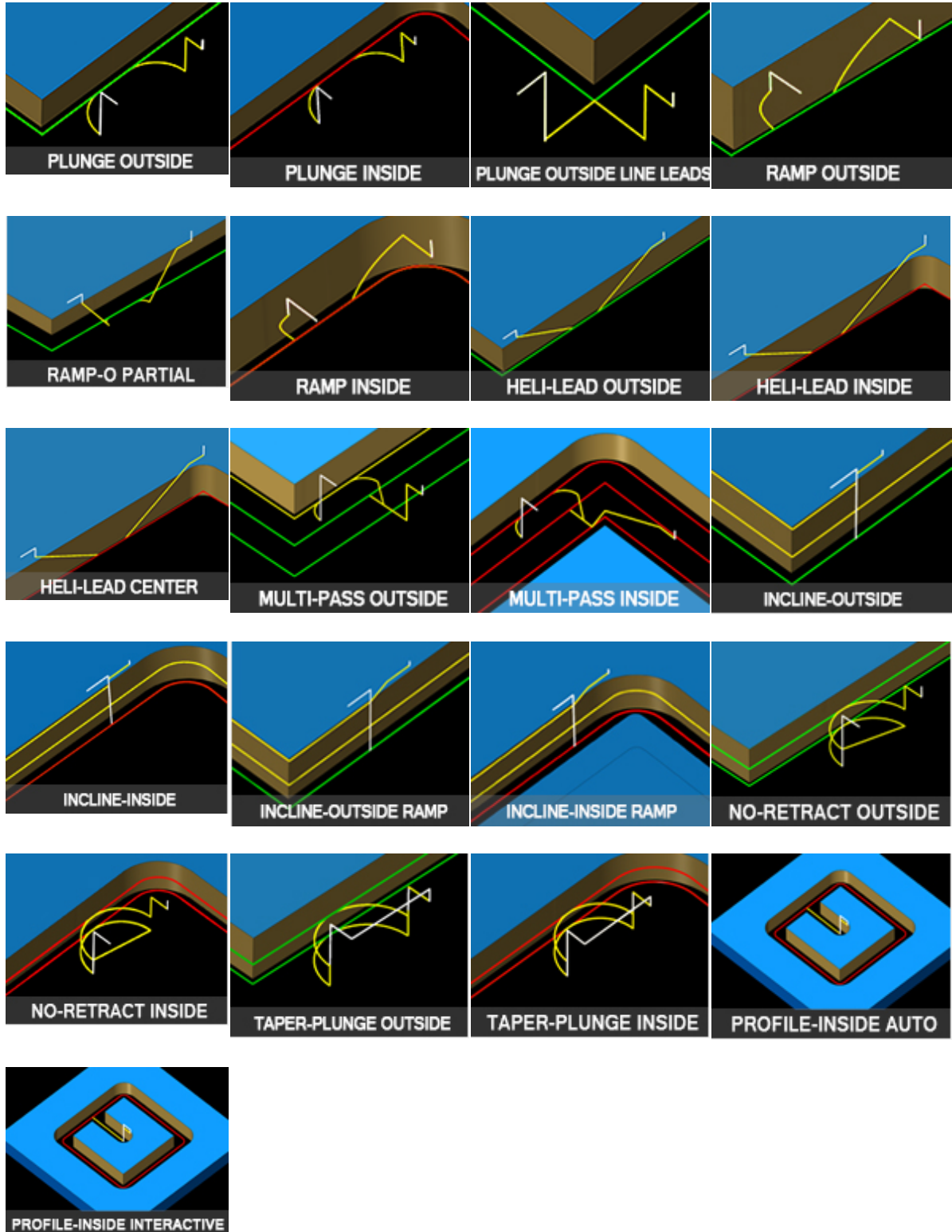
The Cycles are listed according to their functionality. Profiling cycles, Drilling and Hole Cutting cycles, Pocketing cycles, Open Shape cycles, NURBS cycles, and then Specialty cycles.

Underneath the cycle image in the Control Panel, you will see an option for:

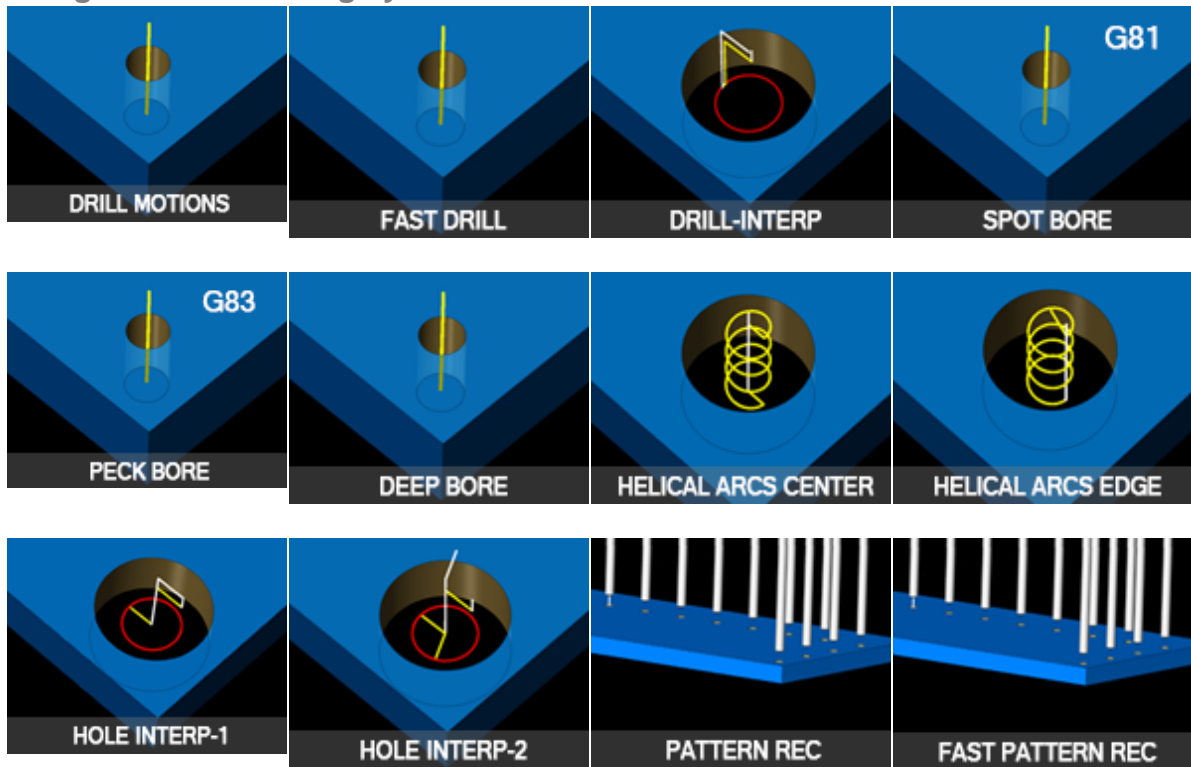


Selecting this option will reset the currently selected cut cycle back to the original default settings.

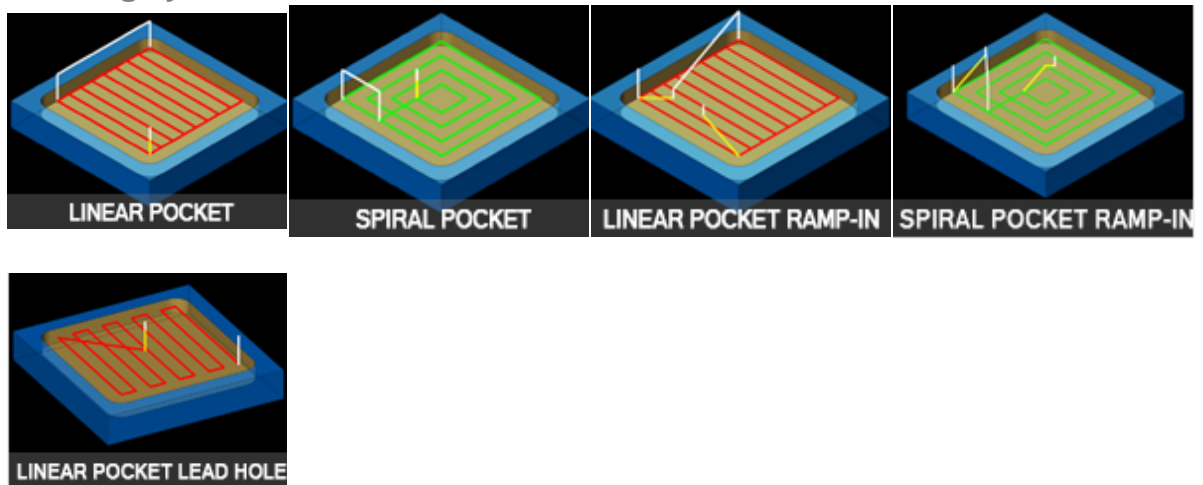
Profiling cycles



Drilling and Hole Cutting cycles



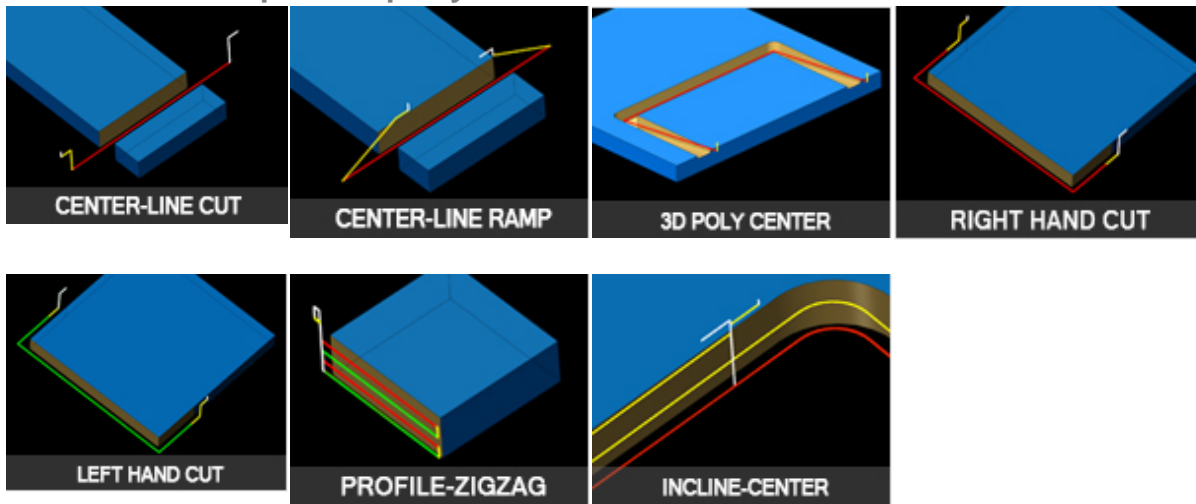
Pocketing cycles



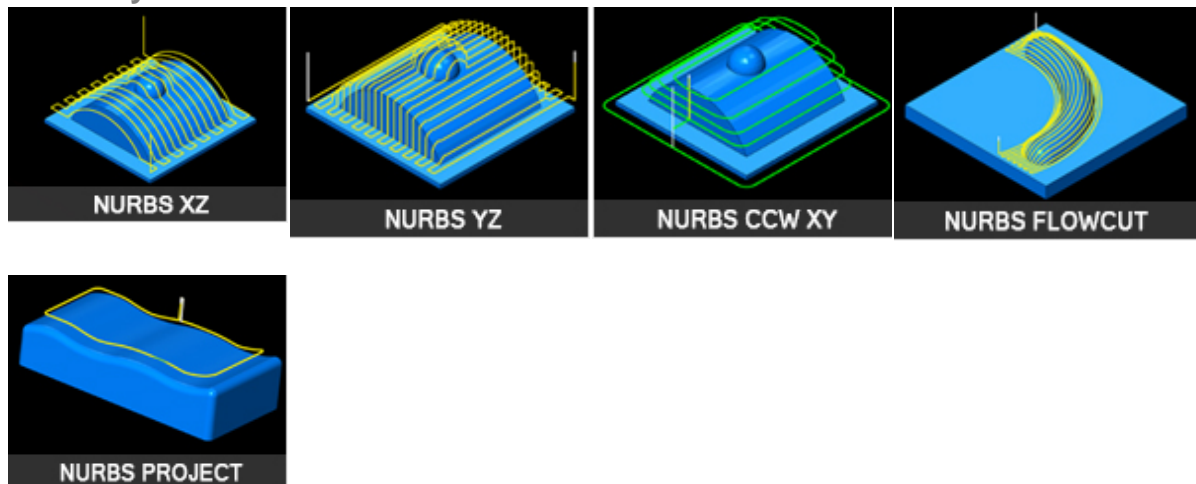
64 Bit Systems Only:



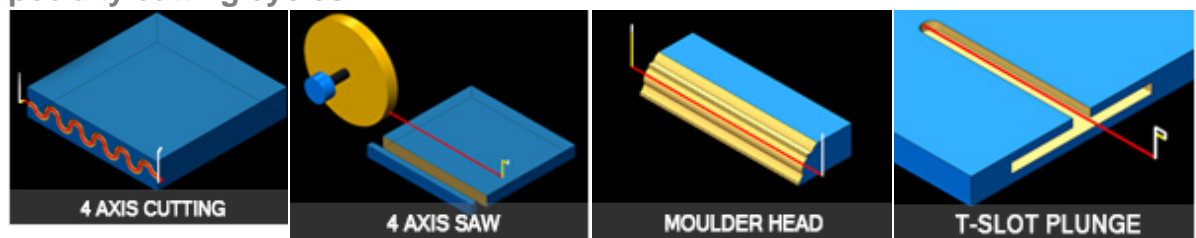
Center Line and Open Shape cycles



NURBS cycles



Specialty cutting cycles





Advanced Machining cycles (Optional)



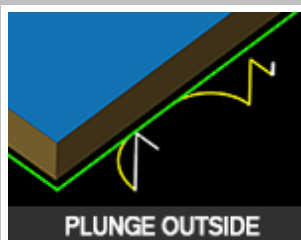
Profiling Cycles

Profiling cycles are Router-CIM cut cycles that are meant to cut a closed shape either on the outside or the inside. These cycles are only meant for closed shapes.

There are several types of profile cuts available to suit a wide variety of tool and material conditions. Each cycle has parameters that control aspects of the cut that you may change in order to manipulate the cut cycle to fit your particular needs.

In the following section, the cut cycles will be explained and each of the parameters available will be described in some detail. At the end of the section there are one or more examples of the use of the cut cycle with the settings used to make the tool path.

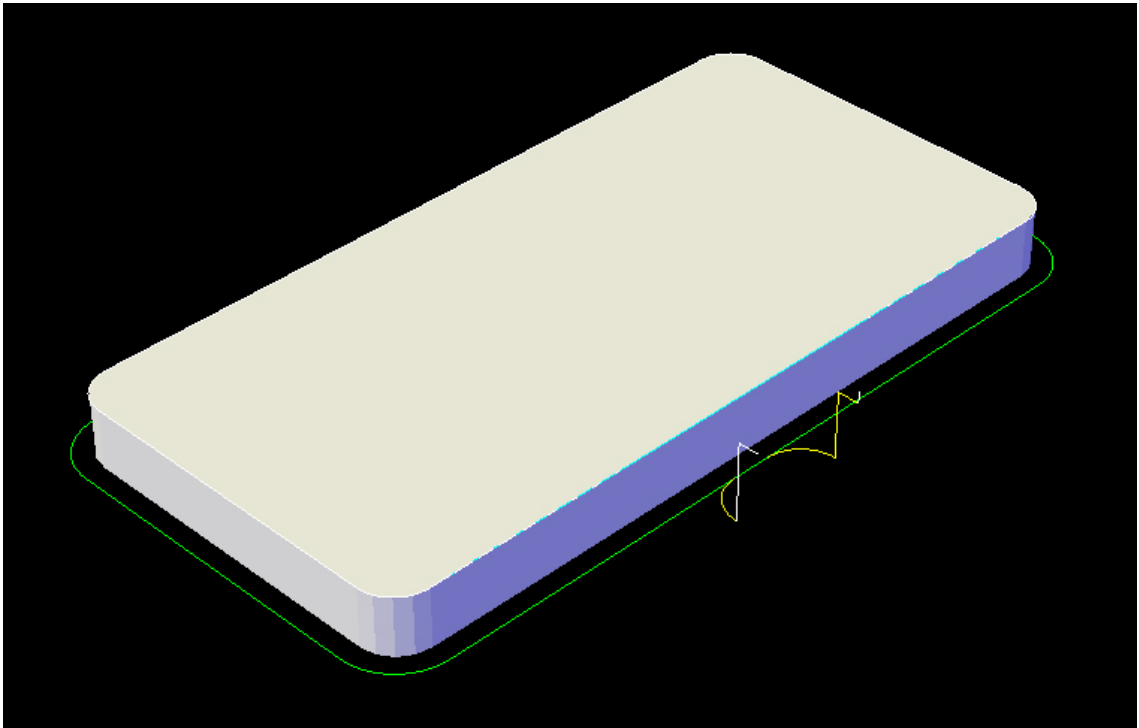
Plunge-Outside



The Plunge-Outside profile cycle will start at the Z height of the Safety Plane, plunge to the point set by Total Cut Depth or Depth per Pass and then make a 90° lead-in arc to the start point, cut the profile shape on the outside back to the start point, then overlap the start point, make a 90° lead-out arc and retract (in rapid) back to the safety plane in Z. The size of the arc lead-in and lead-out are set by the size of the tool selected and may be overridden by the user as necessary. The overlap amount is also able to be modified. There are special parameters to control the lead-in feedrate separately from the cut feedrate, and the lead out move is done in Rapid Traverse (as there is no material there typically).

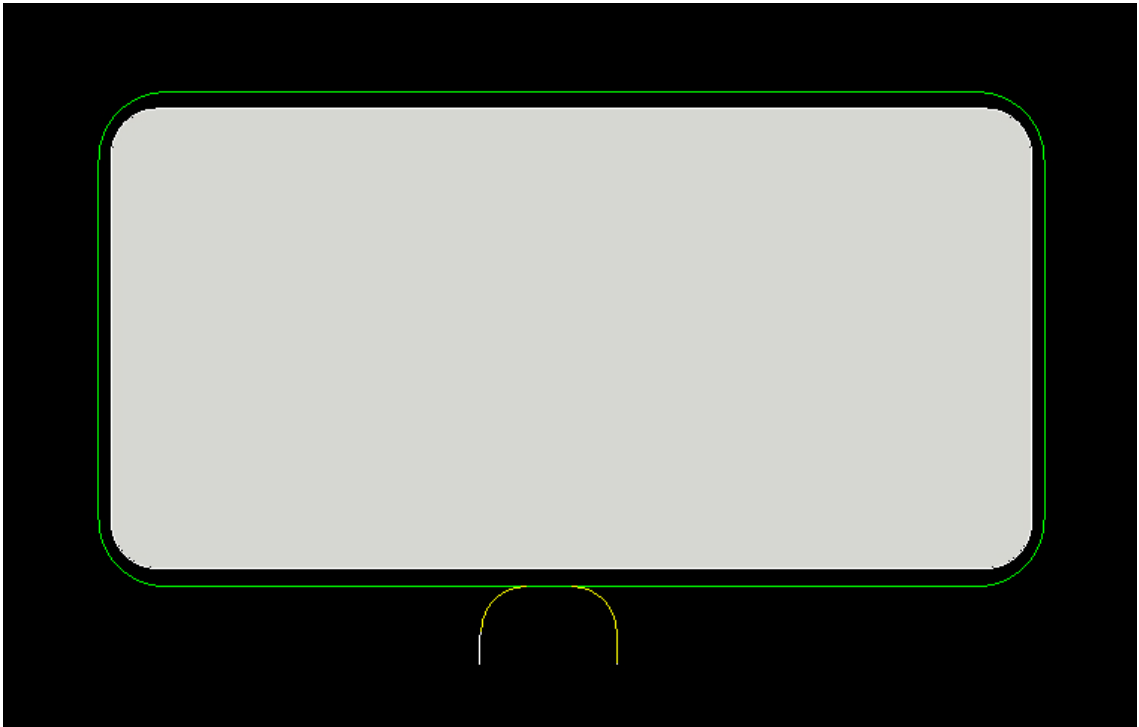
Each of the parameters that control the cycle are explained in more detail below.

In use the cycle looks like this:



Plunge-Outside cycle

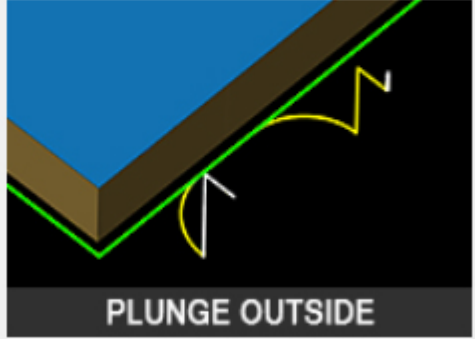
From the top the cycle looks like this:



Plunge-Outside cycle (from above)

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC	Safety Plane	*0.25000
Cut Side	OUTSIDE	Depth Per Pass	1.00000
Cut Direction	CW	Total Cut Depth	
Round Corners	N	Feederate/Spindle Speed	
Lead In	MULTILI	Feederate	350.00000
Lead Out	MULTILO	Spindle Speed	18000.00000
Lead Size	LEADSCL	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Overlap Amount	AUTO	Calculate	
XY Stock Allowance		Before Codes	
Z Stock Allowance		After Codes	
Corner Punch	N	Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Plunge-Outside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit your particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Cornerpunch

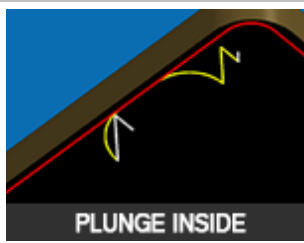
The Cornerpunch feature will find any interior angle (less than 180 degrees) and extend the cut path into the angle by required amount based on the tool size in order for a piece to fit flush within the angle without having to draw the geometry.

Note: The Cornerpunch feature only works with Cutter Compensation set to NO or BOTH.

See the [Cornerpunch](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Plunge-Inside

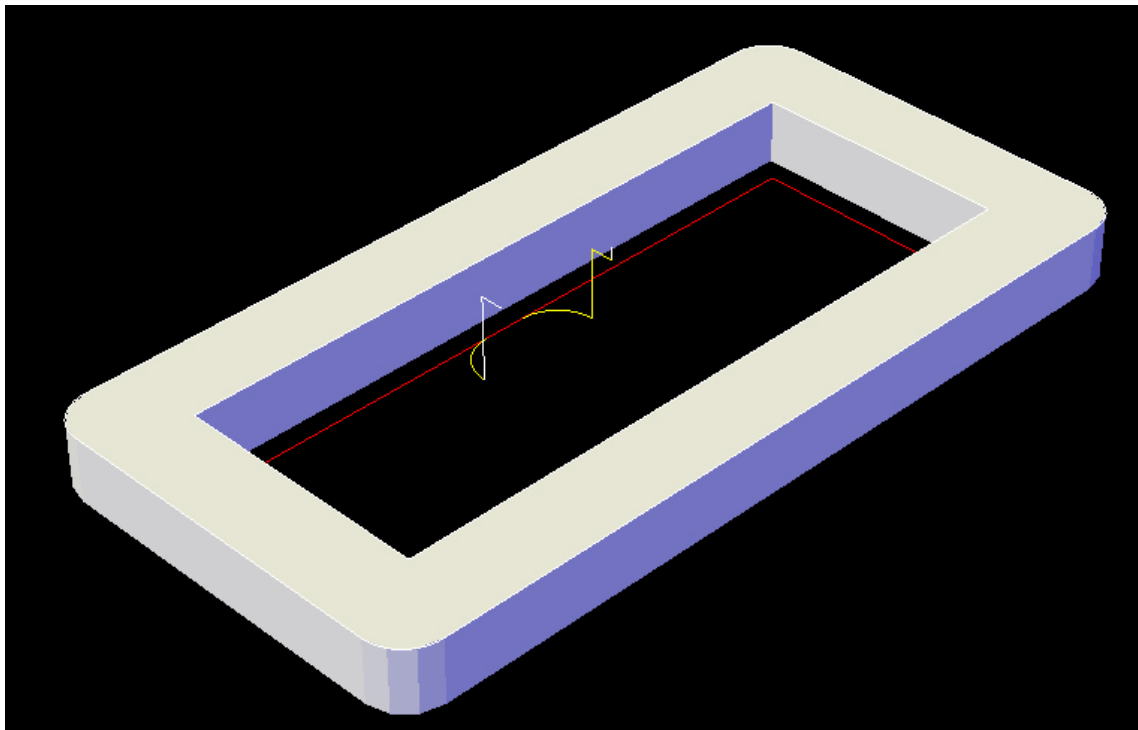


The Plunge-Inside profile cycle will start at the Z height of the Safety Plane, plunge to the point set by Total Cut Depth or Depth per Pass and then make a 90° lead-in arc to the start point, cut the profile shape on the inside back to the start point, then overlap the start point, make a 90° lead-out arc and retract (in rapid) back to the safety plane in Z.

The size of the arc lead-in and lead-out are set by the size of the tool selected and may be overridden by the user as necessary. The overlap amount is also able to be modified. There are special parameters to control the lead-in feedrate separately from the cut feedrate, and the lead out move is done in Rapid Traverse (as there is no material there typically).

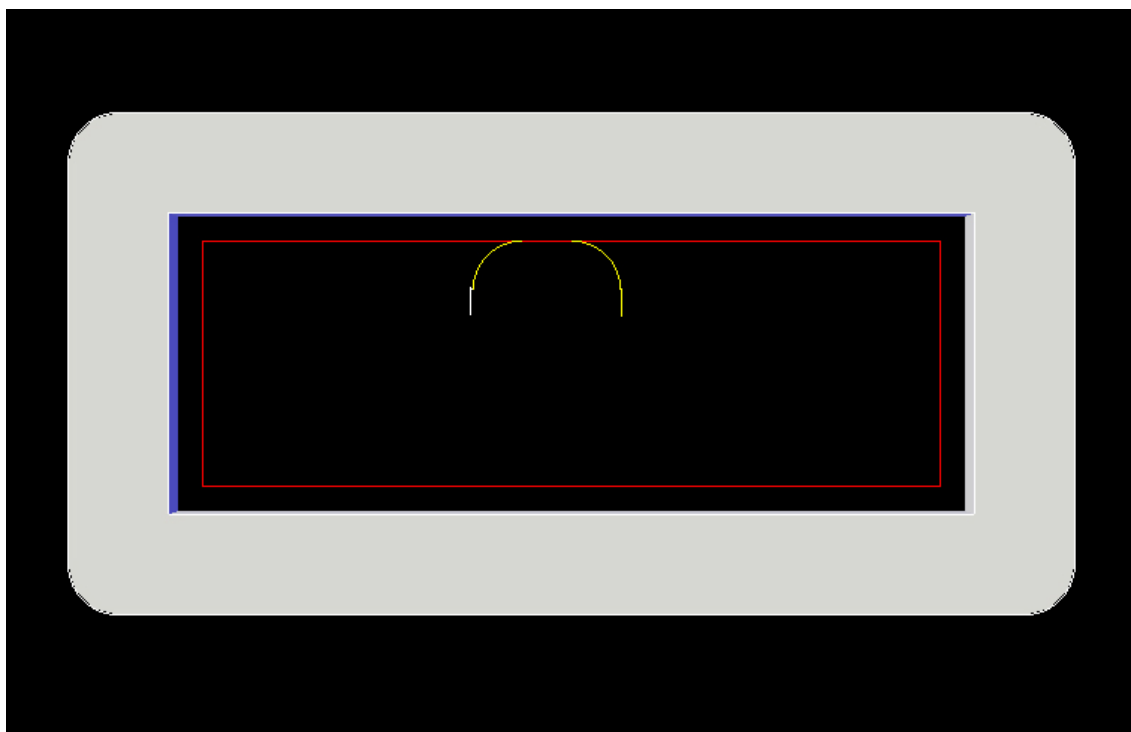
Each of the parameters that control the cycle are explained in more detail below.

In use the cycle looks like this:



Plunge-Inside cycle

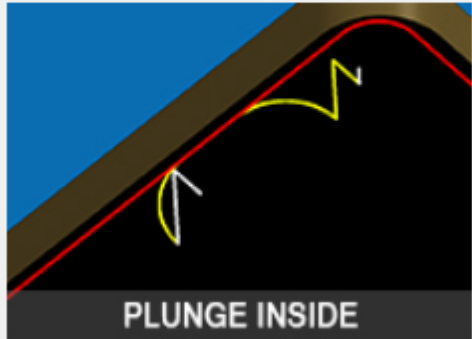
From the top the cycle looks like this:



Plunge-Inside cycle (from above)

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC	Safety Plane	*0.25000
Cut Side	INSIDE	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Round Corners	N	Feederate/Spindle Speed	
Lead In	MULTILI	Feederate	350.00000
Lead Out	MULTILO	Spindle Speed	18000.00000
Lead Size	LEADSCL	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Overlap Amount	AUTO	Calculate	
XY Stock Allowance		Before Codes	
Z Stock Allowance		After Codes	
Corner Punch	N	Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Plunge-Inside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.

Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Cornerpunch

The Cornerpunch feature will find any interior angle (less than 180 degrees) and extend the cut path into the angle by required amount based on the tool size in order for a piece to fit flush within the angle without having to draw the geometry.

Note: The Cornerpunch feature only works with Cutter Compensation set to NO or BOTH.

See the [Cornerpunch](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Plunge-Outside Line Leads



The Plunge-Outside Line Leads cycle will start at the Safety Plane, plunge to the Total Cut Depth or Depth per Pass and then make a lead-in move tangent to the first cut, then follow the entire shape given, and make a lead-out tangent to the last move, and finally retract back to the Safety Plane. There is no overlap by default.

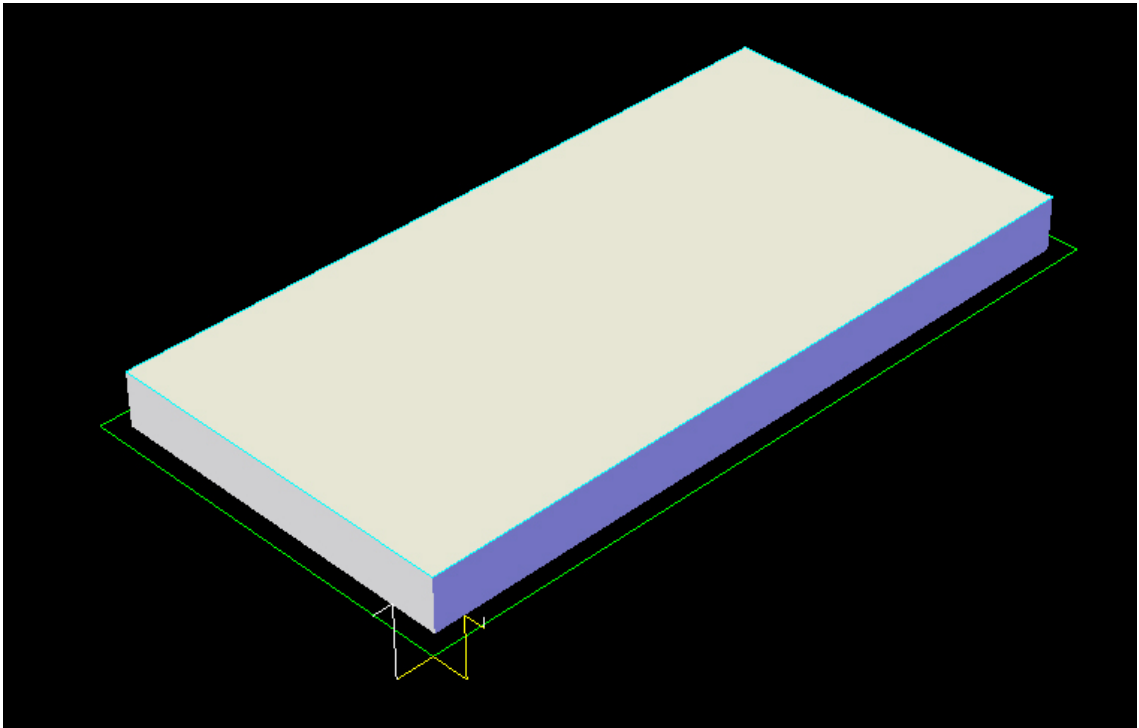
This cycle is used most often on outside shapes where the tool is plunged to full depth outside of the part, then lead into the shape to cut the outside and led out away from the part before retracting. The size of the lead-in and lead-out may be controlled by the user, as well as the geometry of the lead-in or lead-out as desired.

The Plunge-Outside Line Leads cycle is a somewhat generic profile cutting cycle in that the parameters used for this cut cycle are able to be modified to create many types of profile cuts. Each of the parameters shown in the section below will be explained in detail for a better explanation of their capabilities.

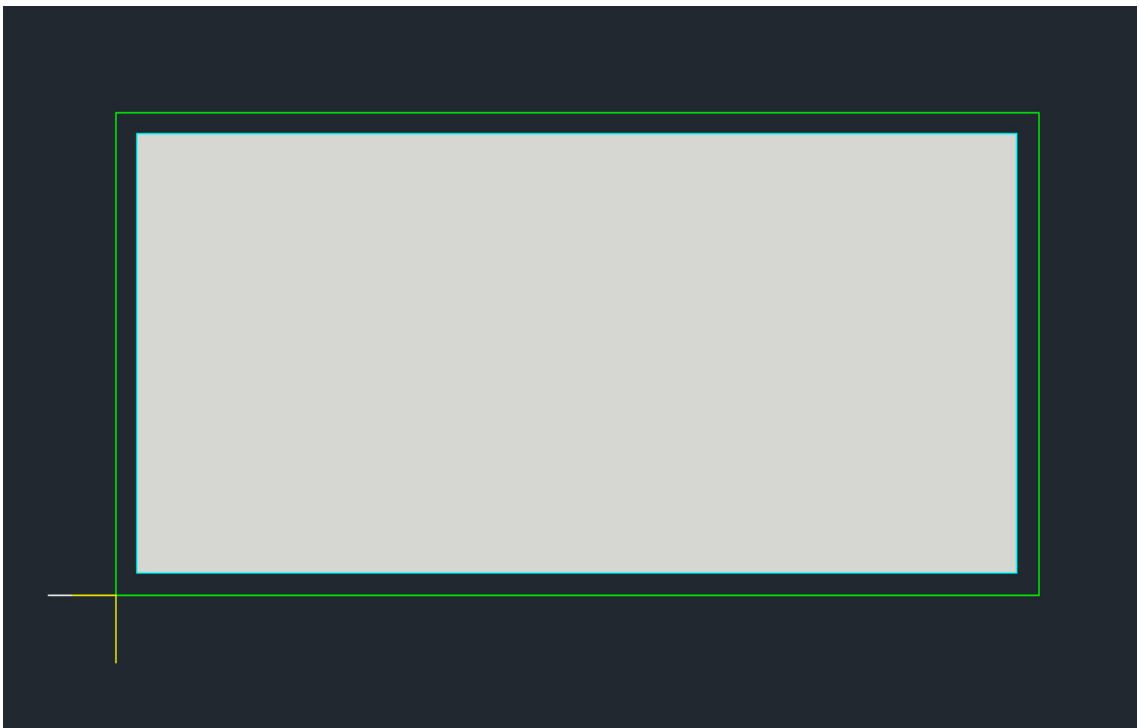
Remember that this cycle has no overlap, so if the Start Point is along an edge you will see a mark on the part where the cutter started and ended. Use this cycle for cutting open shapes, because it allows the user to modify the Lead-In, Lead-Out, Cut Side, Cut Direction, Offset Amount, and Lead Size parameters to fit a specific cutting task.

This cycle is used mostly for cutting with shaper or profile tools, when you want to lower the cutter to full depth away from the finished part, and allow the cutter to rotate up to speed before engaging the finished part.

In use the cycle looks like this:




Plunge-Outside Line Leads cycle



Plunge-Outside Line Leads (from above)

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC	Safety Plane	*0.25000
Cut Side	OUTSIDE	Depth Per Pass	1.00000
Cut Direction	CW	Total Cut Depth	
Round Corners	N	Feedrate/Spindle Speed	
Lead In	"LNTLI"	Feedrate	350.00000
Lead Out	"LNTLO"	Spindle Speed	18000.00000
Lead Size	LEADSCL	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Overlap Amount	AUTO	Calculate	
XY Stock Allowance		Before Codes	
Z Stock Allowance		After Codes	
Corner Punch	N	Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Plunge-Outside Line Leads cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits.

By default this cycle will use LNTLI (Line Tangent Lead In) but can be changed as needed.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits.

By default this cycle will use LNTLO (Line Tangent Lead Out) but can be changed as needed.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Cornerpunch

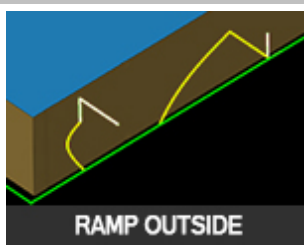
The Cornerpunch feature will find any interior angle (less than 180 degrees) and extend the cut path into the angle by required amount based on the tool size in order for a piece to fit flush within the angle without having to draw the geometry.

Note: The Cornerpunch feature only works with Cutter Compensation set to NO or BOTH.

See the [Cornerpunch](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Ramp-Outside

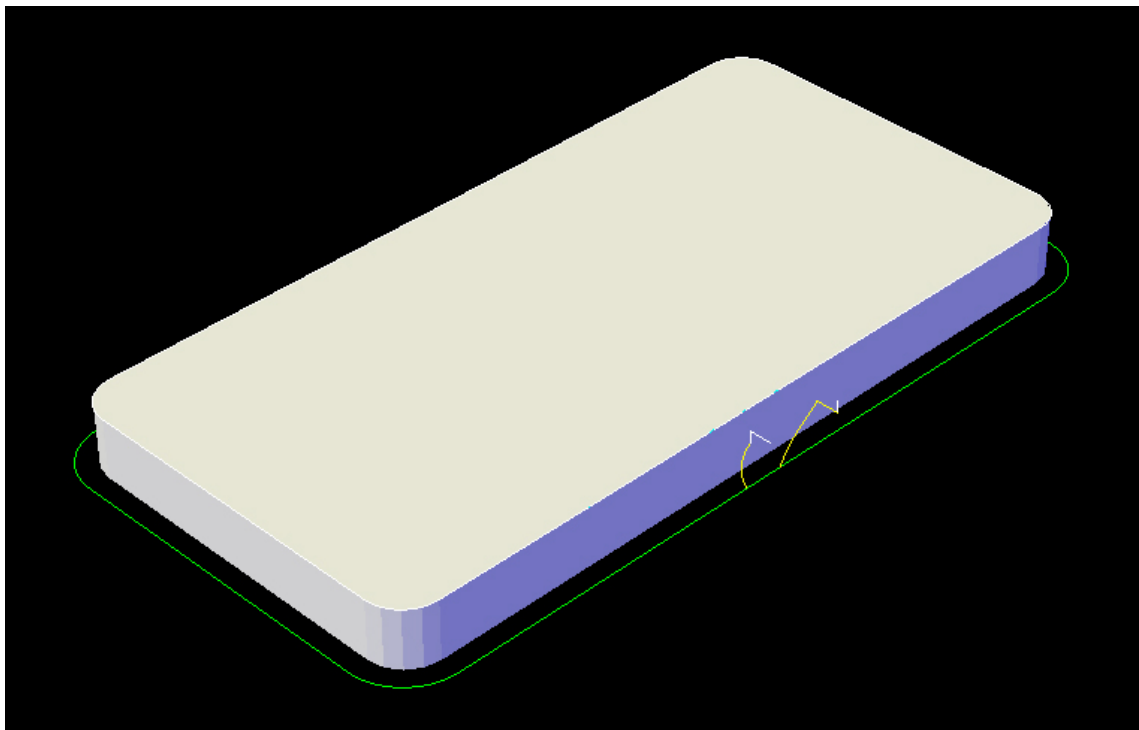


Ramp Cutting cycles allow you to have a helical lead-in and lead-out from a point perpendicular to the part.

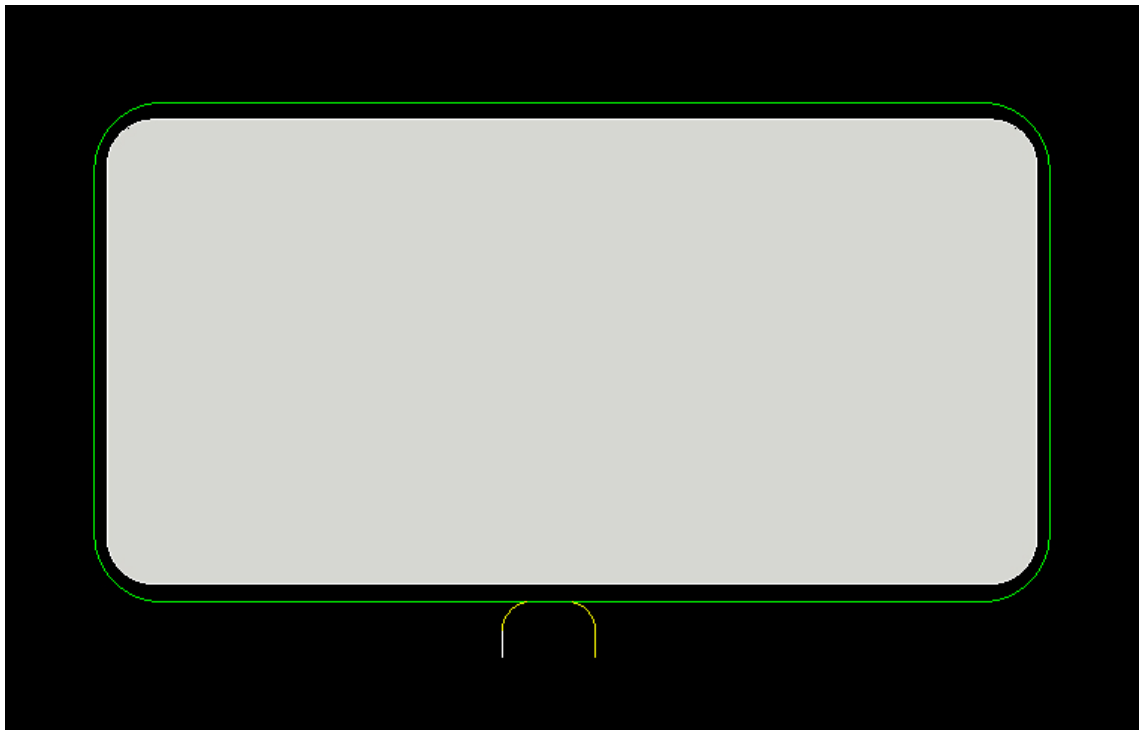
This cycle is especially useful to save on tool wear, and in circumstances where the cutter cannot plunge into the material.

Ramp Outside and Ramp Inside only work on closed shapes, and produce cuts with an offset to the outside or inside of the geometry according to the cycle.

The tool will start above the part at the Safety Plane, and then make a 90° arc, ramping lead-in to the start point. Once at depth the cutter will follow the shape on the outside and cut back to the start point, overlap the start point and then make a 90° arc, ramping lead-out back up to the Safety Plane.



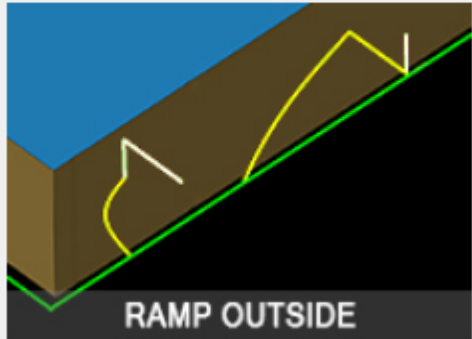
Ramp-Outside cycle



Ramp-Outside (from above)

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:

Cycle Information		Status Information	
Offset Dim	OFFSZ	Safety Plane	*0.25000
Cut Side	OUTSIDE	Depth Per Pass	1.00000
Cut Direction	CW	Total Cut Depth	
Round Corners	N	Feedrate/Spindle Speed	
Lead In	MULTILI	Feedrate	350.00000
Lead Out	MULTILO	Spindle Speed	18000.00000
Lead Size	!tr*	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Ramp Amount	NONE	Calculate	
Overlap Amount	AUTO	Before Codes	
XY Stock Allowance		After Codes	
Z Stock Allowance		Oscillation Amount	0.00000
Corner Punch	N	Sort By Rank #	
			
		Reset Cycle Settings to Default	

Ramp-Outside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Ramp Amount

The Ramp Amount is the distance during the lead-in that the cutter spends in the ramp to the Total Depth of the Cut.

See the [Ramp Amount](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Cornerpunch

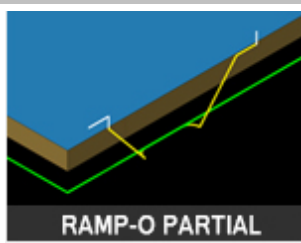
The Cornerpunch feature will find any interior angle (less than 180 degrees) and extend the cut path into the angle by required amount based on the tool size in order for a piece to fit flush within the angle without having to draw the geometry.

Note: The Cornerpunch feature only works with Cutter Compensation set to NO or BOTH.

See the [Cornerpunch](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

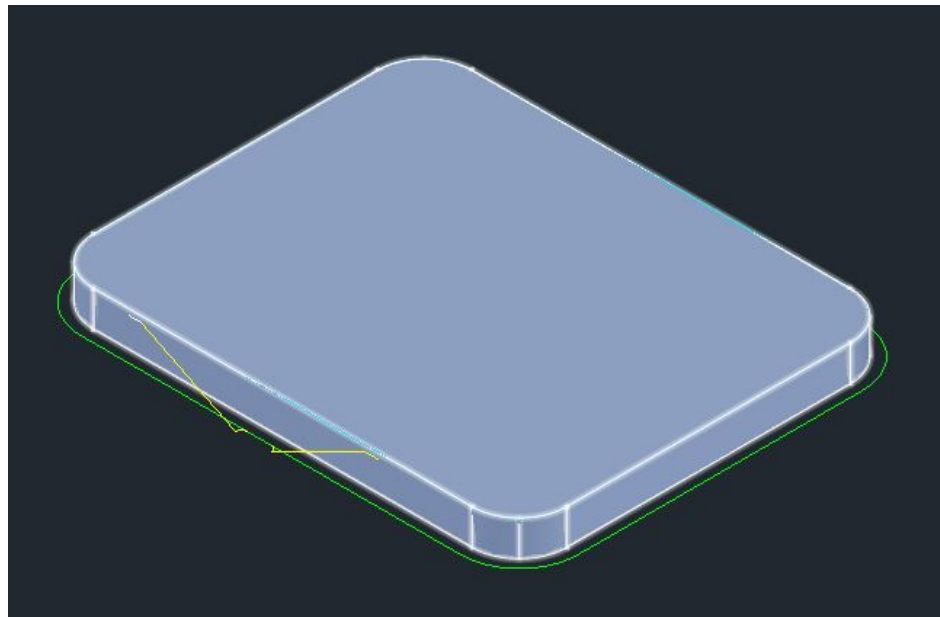
Ramp-Outside Partial



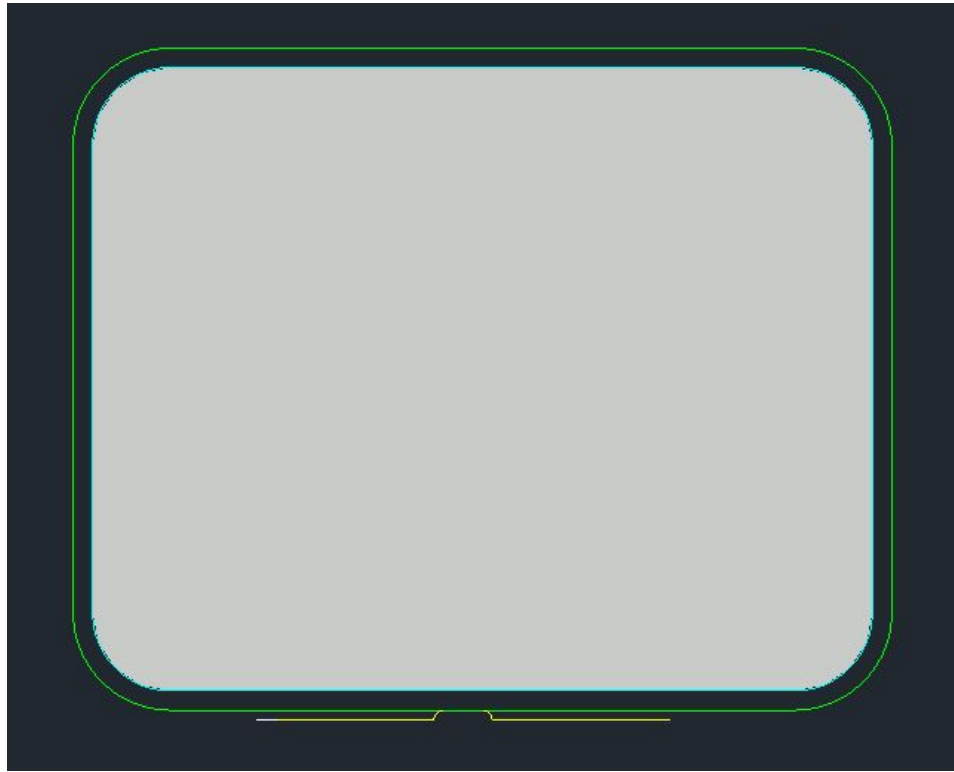
Ramp Outside Partial cutting cycle allow you to have a helical lead-in and lead-out from a point perpendicular to the part. This cycle is especially useful to save on tool wear, and in circumstances where the cutter cannot plunge into the material. This cycle will also let the user define an offset so that the ramp lead-in and lead-out will be at the Total Cut Depth prior to moving into the part edge helping prevent deformations on the bottom of the parts.

Ramp Outside Partial will only work on closed shapes, and produce cuts with an offset to the outside of the geometry according to the cycle.

The tool will start above the part at the Safety Plane, and then make a linear ramp lead-in to the Total Cut Depth defined and will be offset from the edge of the part by the LeadOffset parameter. Once at depth, the cutter will follow a 90° arc to complete the lead-in ending up at the edge of the part and cut back to the start point, overlap the start point, make a 90° arc away from the part edge and then make a linear ramp lead-out back up to the Safety Plane.



Ramp-Outside Partial cycle

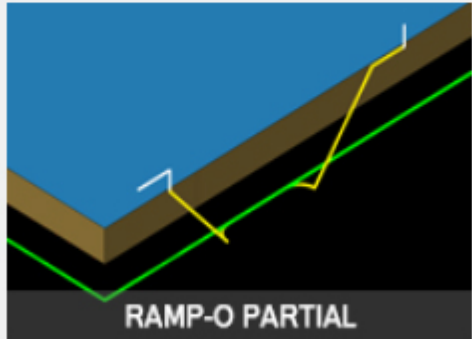


Ramp-Outside Partial (from above)

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:

Note: In order to use this cycle, LeadOffset and Ramp Amt. need to be properly defined.

Cycle Information		Status Information	
Offset Dim	OFFSZ	Safety Plane	*0.25000
Cut Side	OUTSIDE	Depth Per Pass	1.00000
Cut Direction	CW	Total Cut Depth	
Round Corners	N	Feedrate/Spindle Speed	
Lead In	MULTILIPART	Feedrate	350.00000
Lead Out	MULTILOPART	Spindle Speed	18000.00000
Lead Size	1.0	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Lead Offset	.125	Calculate	
Ramp Amount	NONE	Before Codes	
Overlap Amount	AUTO	After Codes	
XY Stock Allowance		Oscillation Amount	0.00000
Z Stock Allowance		Sort By Rank #	
Corner Punch	N		
		Reset Cycle Settings to Default	

Ramp-Outside Partial cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Lead Offset

This sets the amount of offset used when the cycle has completed the linear ramp lead-in and starts the 90° arc into the part edge. This field must be a positive number. The larger the number used, the larger the 90° arc will be.

Ramp Amount

The Ramp Amount is the distance during the lead-in that the cutter spends in the ramp to the Total Depth of the Cut.

See the [Ramp Amount](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Cornerpunch

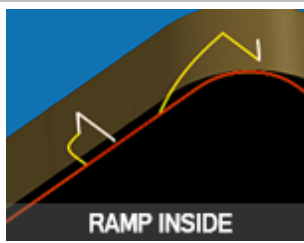
The Cornerpunch feature will find any interior angle (less than 180 degrees) and extend the cut path into the angle by required amount based on the tool size in order for a piece to fit flush within the angle without having to draw the geometry.

Note: The Cornerpunch feature only works with Cutter Compensation set to NO or BOTH.

See the [Cornerpunch](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Ramp-Inside

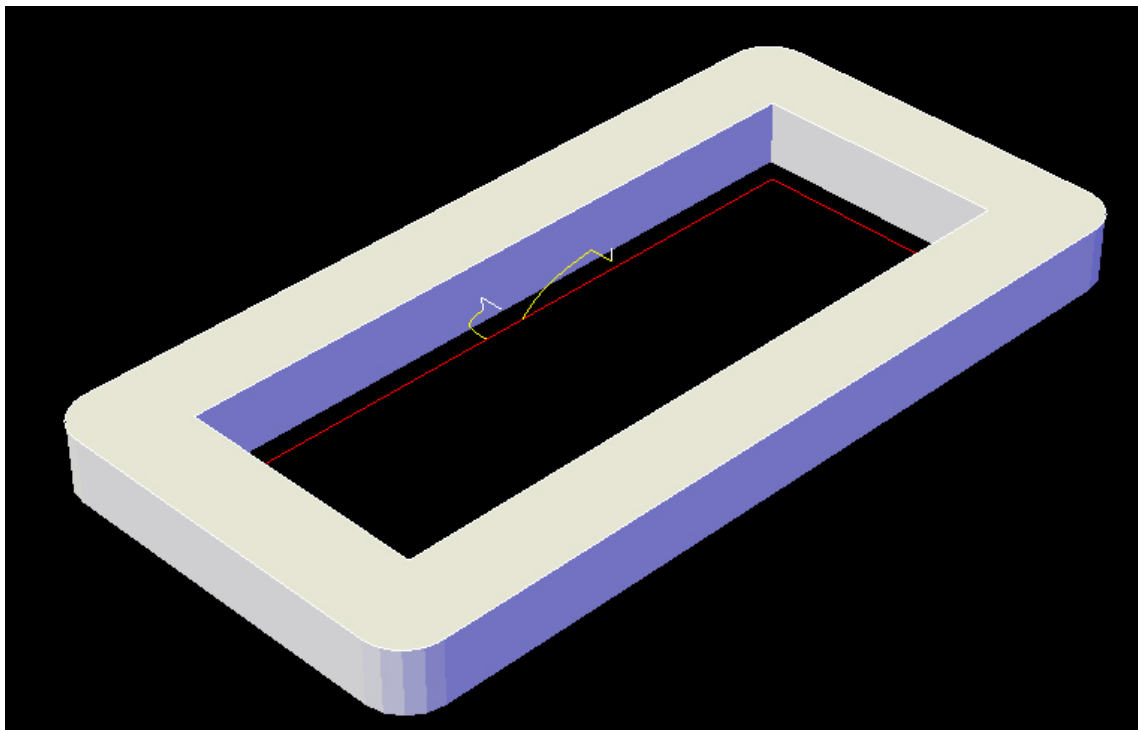


Ramp Cutting cycles allow you to have a helical lead-in and lead-out from a point perpendicular to the part.

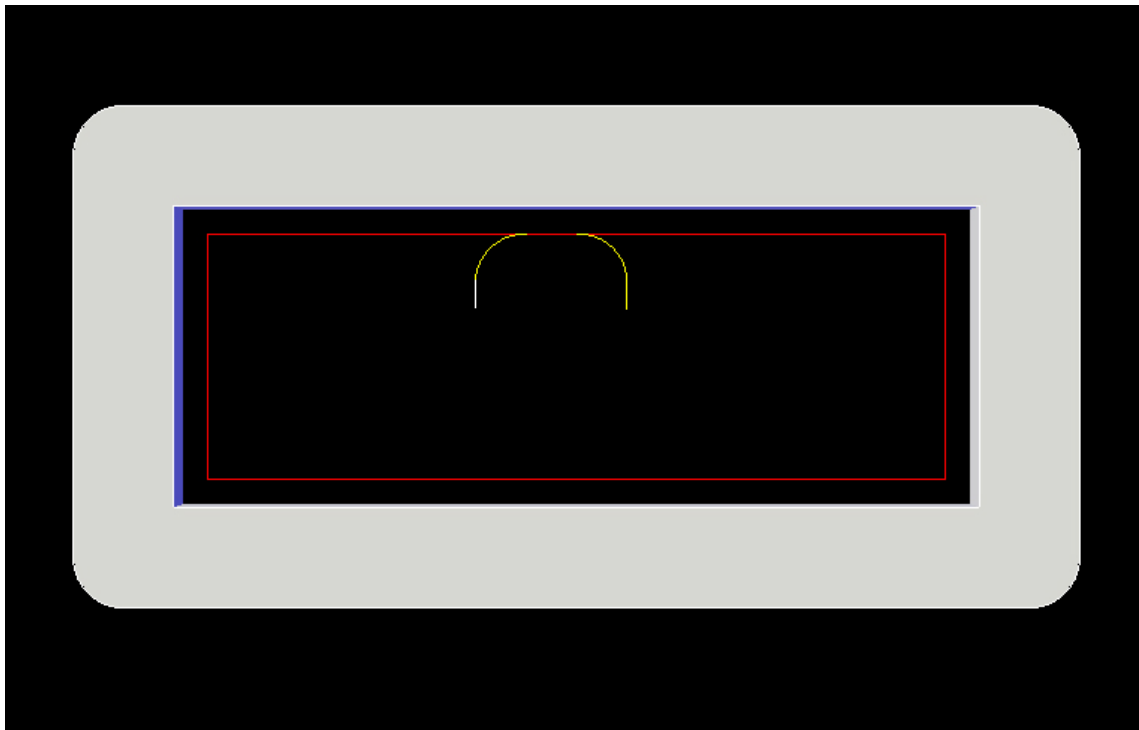
This cycle is especially useful to save on tool wear, and in circumstances where the cutter cannot plunge into the material.

Ramp Outside and Ramp Inside only work on closed shapes, and produce cuts with an offset to the outside or inside of the geometry according to the cycle.

The tool will start above the part at the Safety Plane, and then make a 90° arc, ramping lead-in to the start point. Once at depth the cutter will follow the shape on the outside and cut back to the start point, overlap the start point and then make a 90° arc, ramping lead-out back up to the Safety Plane.



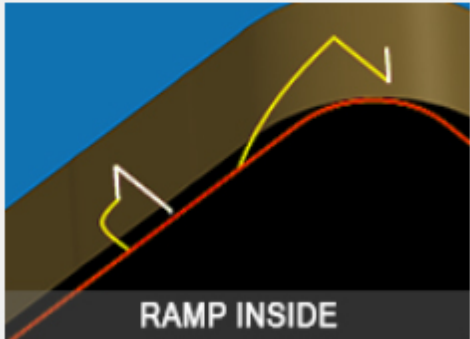
Ramp-Inside cycle



Ramp-Inside (from above)

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:

Cycle Information		Status Information	
Offset Dim	OFFSZ	Safety Plane	*0.25000
Cut Side	INSIDE	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Round Corners	N	Feedrate/Spindle Speed	
Lead In	MULTILI	Feedrate	350.00000
Lead Out	MULTILO	Spindle Speed	18000.00000
Lead Size	!*tr*	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Ramp Amount	NONE	Calculate	
Overlap Amount	AUTO	Before Codes	
XY Stock Allowance		After Codes	
Z Stock Allowance		Oscillation Amount	0.00000
Corner Punch	N	Sort By Rank #	
			
		Reset Cycle Settings to Default	

Ramp-Inside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Ramp Amount

The Ramp Amount is the distance during the lead-in that the cutter spends in the ramp to the Total Depth of the Cut.

See the [Ramp Amount](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Cornerpunch

The Cornerpunch feature will find any interior angle (less than 180 degrees) and extend the cut path into the angle by required amount based on the tool size in order for a piece to fit flush within the angle without having to draw the geometry.

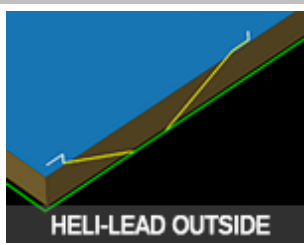
Note: The Cornerpunch feature only works with Cutter Compensation set to NO or BOTH.

See the [Cornerpunch](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the**

toolpath and NC Code carefully before running your machine tool if you change these default settings.

Heli-Lead Outside

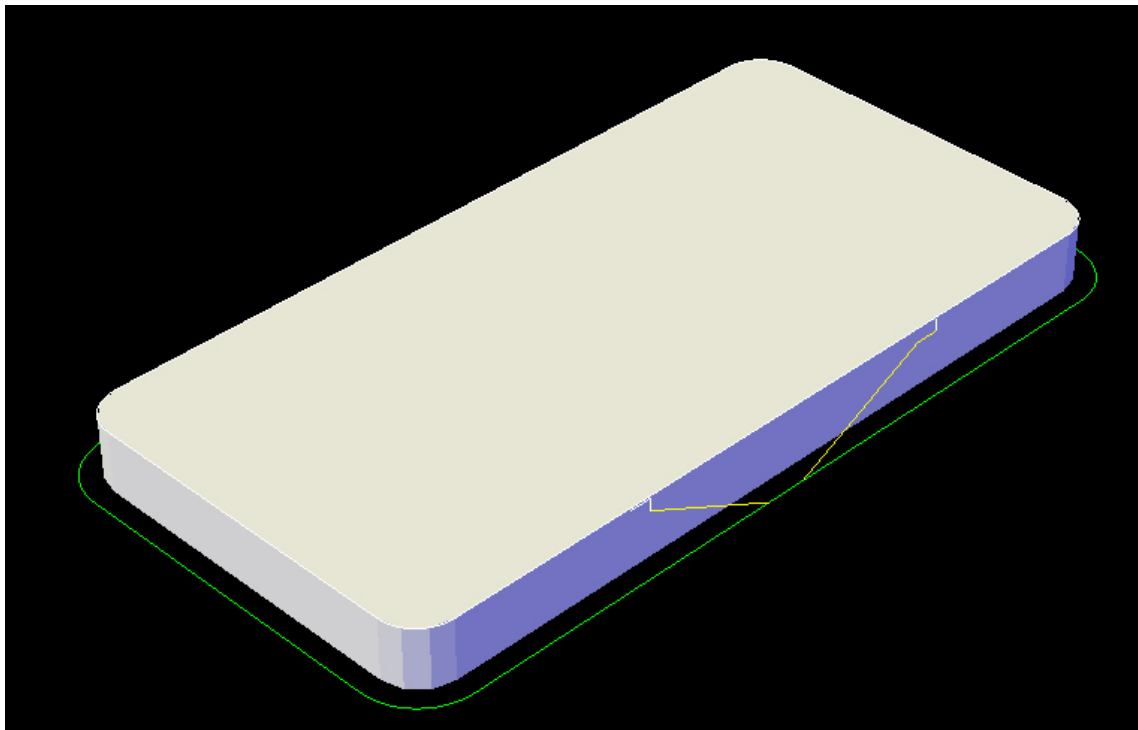


Heli-Lead Cutting cycles allow you to have a contour following lead-in and lead-out. This means that on a straight cut, the lead in and lead out would be a ramp (XZ or YZ motion) and on a non-straight shape the lead-in and lead-out will follow the contour of the shape, no matter the complexity.

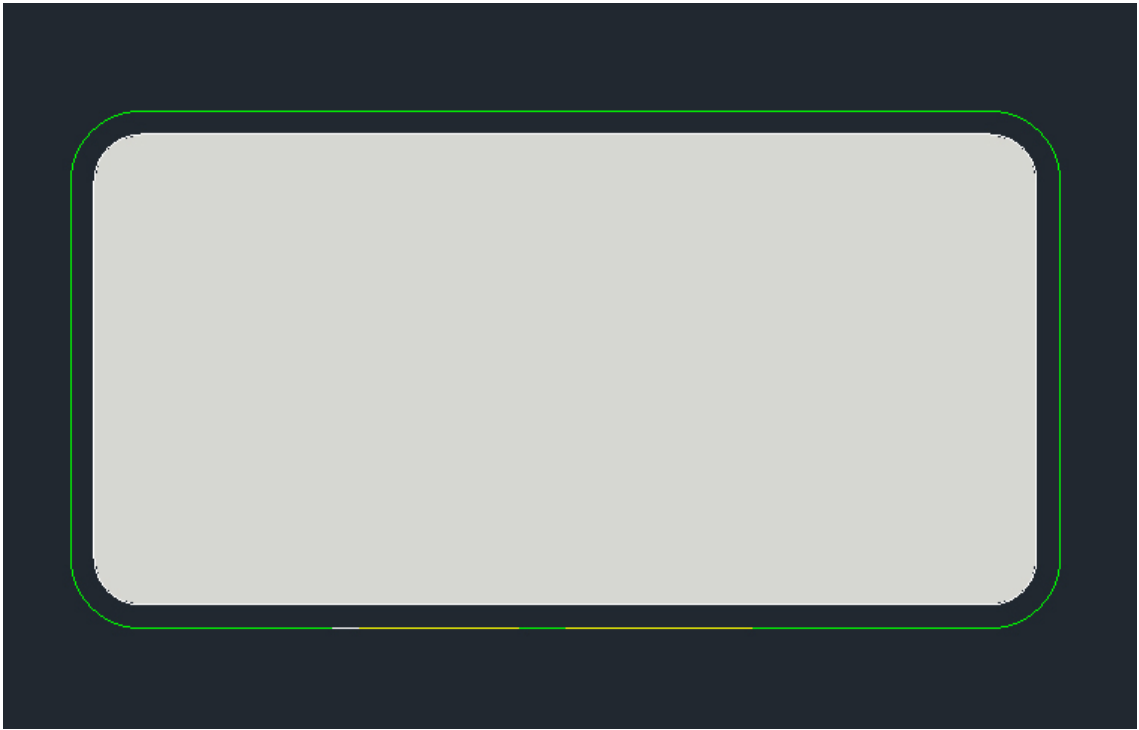
These cycles are especially useful when cutting interior shapes or parts in a nest; the scrap will not move during the lead-out. These cycles only work on closed shapes.

Typically, Heli-Lead Outside will start above the part at the Safety Plane and then ramp into the part along the geoshape, following the contour, until it reaches full cut depth. The cycle will follow the shape back to the start point, overlap by the tool diameter, then ramp out of the shape, again following the contour, back up to the Safety Plane.

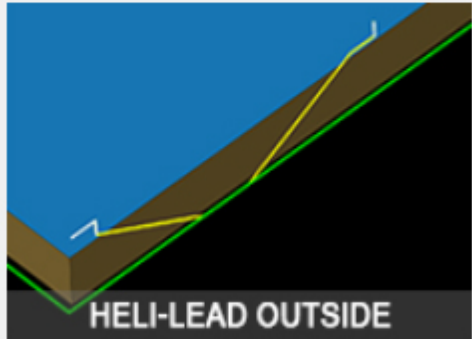
This is one of the most commonly used cycles in Router-CIM, because they produce no scrap - the leads are always tangent to the part geometry.



Heli-Lead-Outside cycle



Heli-Lead-Outside cycle (from above)

Cycle Information		Status Information	
Offset Dim	OFFSZ	Safety Plane	*0.25000
Cut Side	OUTSIDE	Depth Per Pass	1.00000
Cut Direction	CW	Total Cut Depth	
Round Corners	N	Feedrate/Spindle Speed	
Lead Size	0.0	Feedrate	350.00000
Lead Angle	N	Spindle Speed	18000.00000
Lead Ratio		Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Overlap Amount	AUTO	Calculate	
XY Stock Allowance		Before Codes	
Z Stock Allowance		After Codes	
Corner Punch	N	Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Heli-Lead-Outside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (OFFSZ) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the [Lead Angle](#) section for more information.

Lead Ratio

Lead Ratio determines the angle of the ramp in Z during the lead in and lead out. You can specify the Lead Ratio as a number that reflects the percentage of the angle from its default. That means that if you want a lead that is twice the normal ramp length (shallower angle) enter 2. If you want a lead that is steeper than the default, enter .5.

See the [Lead Ratio](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.

Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Cornerpunch

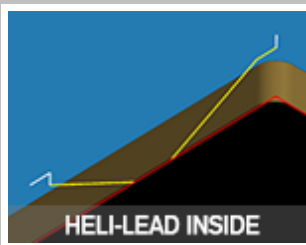
The Cornerpunch feature will find any interior angle (less than 180 degrees) and extend the cut path into the angle by required amount based on the tool size in order for a piece to fit flush within the angle without having to draw the geometry.

Note: The Cornerpunch feature only works with Cutter Compensation set to NO or BOTH.

See the [Cornerpunch](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Heli-Lead-Inside

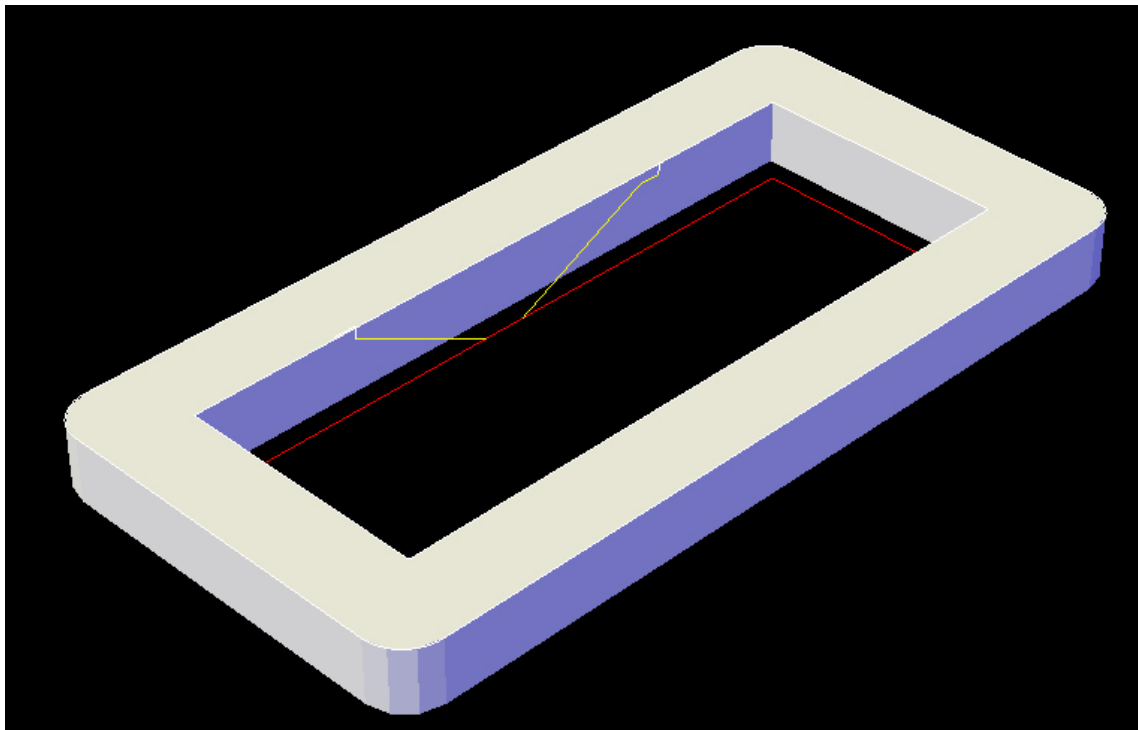


Heli-Lead Cutting cycles allow you to have a contour following lead-in and lead-out. This means that on a straight cut, the lead in and lead out would be a ramp (XZ or YZ motion) and on a non-straight shape the lead-in and lead-out will follow the contour of the shape, no matter the complexity.

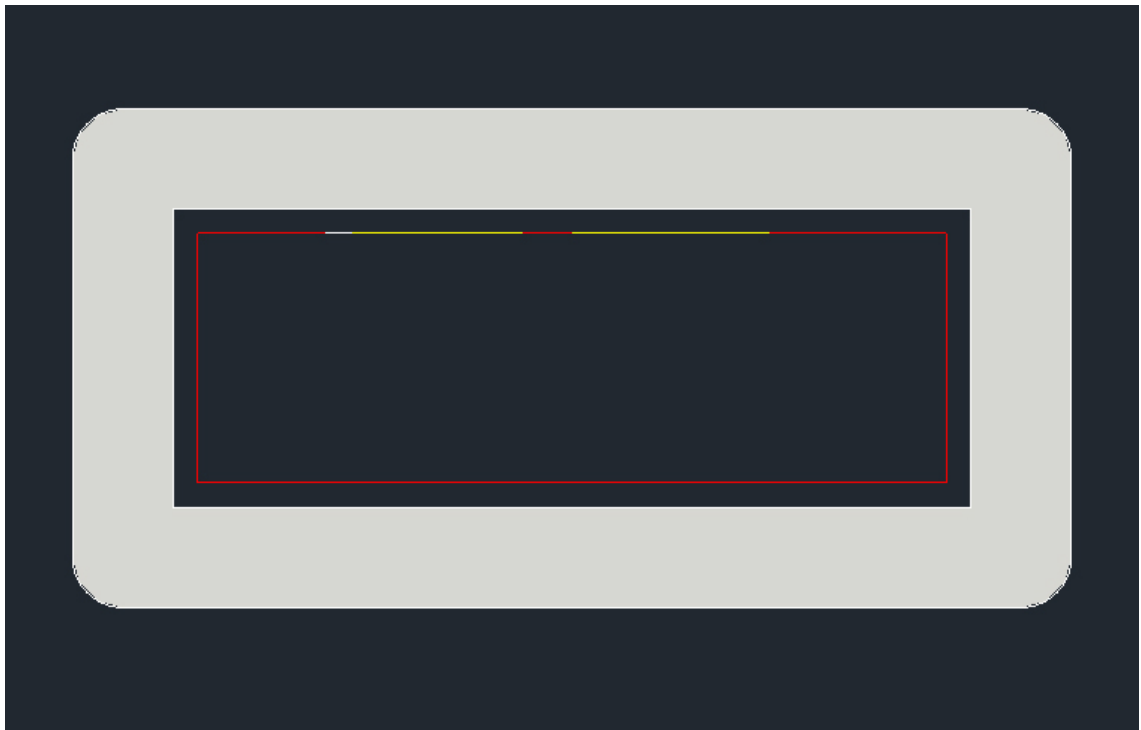
These cycles are especially useful when cutting interior shapes or parts in a nest; the scrap will not move during the lead-out. These cycles only work on closed shapes.

Typically, Heli-Lead Inside will start above the part at the Safety Plane and then ramp into the part along the geoshape, following the contour, until it reaches full cut depth. The cycle will follow the shape back to the start point, overlap by the tool diameter, then ramp out of the shape, again following the contour, back up to the Safety Plane.

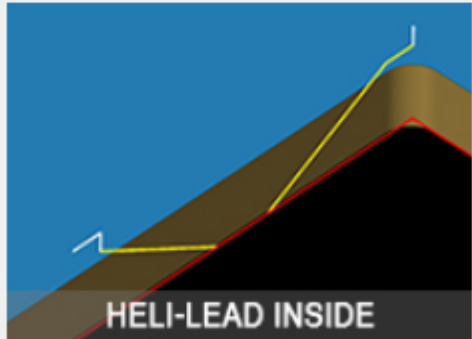
This is one of the most commonly used cycles in Router-CIM, because they produce no scrap - the leads are always tangent to the part geometry.



Heli-Lead-Inside cycle



Heli-Lead-Inside cycle (from above)

Cycle Information		Status Information	
Offset Dim	OFFSZ	Safety Plane	*0.25000
Cut Side	INSIDE	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Round Corners	N	Feedrate/Spindle Speed	
Lead Size	0.0	Feedrate	350.00000
Lead Angle	N	Spindle Speed	18000.00000
Lead Ratio		Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Overlap Amount	AUTO	Calculate	
XY Stock Allowance		Before Codes	
Z Stock Allowance		After Codes	
Corner Punch	N	Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Heli-Lead-Inside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (OFFSZ) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the [Lead Angle](#) section for more information.

Lead Ratio

Lead Ratio determines the angle of the ramp in Z during the lead in and lead out. You can specify the Lead Ratio as a number that reflects the percentage of the angle from its default. That means that if you want a lead that is twice the normal ramp length (shallower angle) enter 2. If you want a lead that is steeper than the default, enter .5.

See the [Lead Ratio](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.

Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Cornerpunch

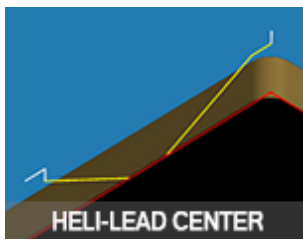
The Cornerpunch feature will find any interior angle (less than 180 degrees) and extend the cut path into the angle by required amount based on the tool size in order for a piece to fit flush within the angle without having to draw the geometry.

Note: The Cornerpunch feature only works with Cutter Compensation set to NO or BOTH.

See the [Cornerpunch](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Heli-Lead Center

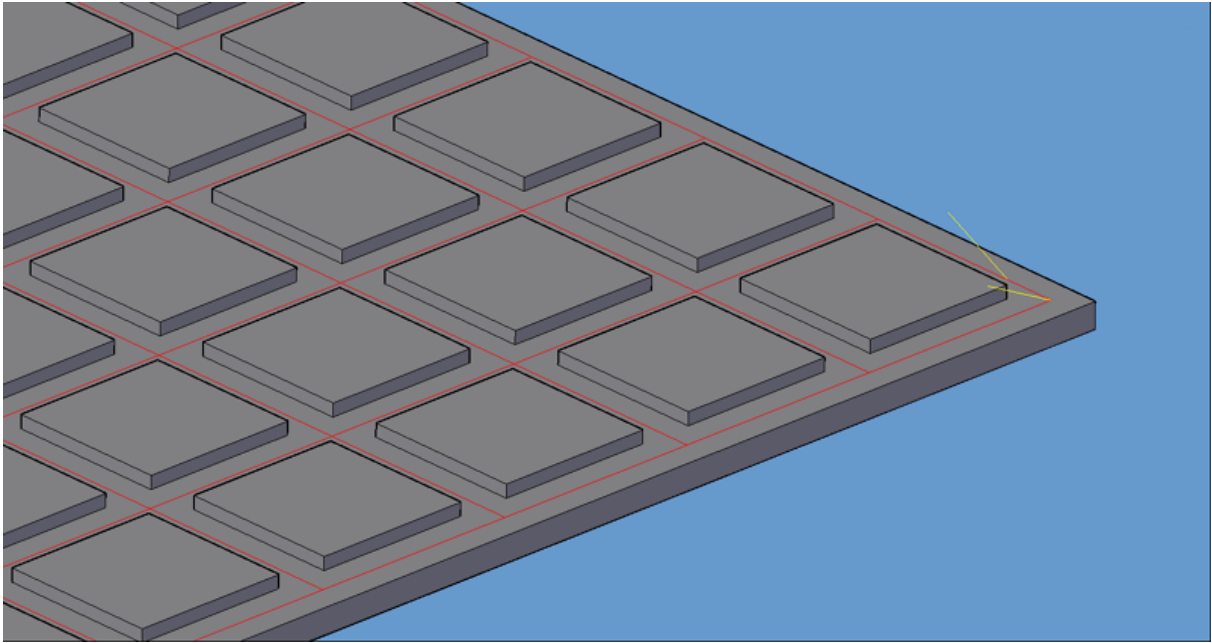


Heli-Lead Cutting cycles allow you to have a contour following lead-in and lead-out. This means that on a straight cut, the lead in and lead out would be a ramp (XZ or YZ motion) and on a non-straight shape the lead-in and lead-out will follow the contour of the shape, no matter the complexity.

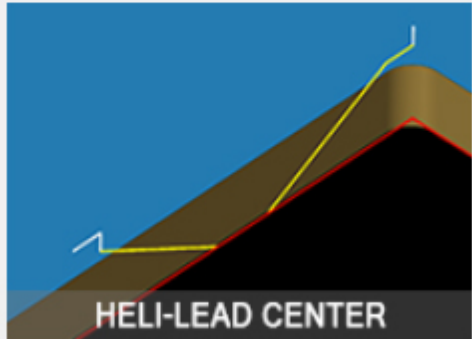
This cycle is especially useful when cutting stay-down tool path nests.

Typically, Heli-Lead Center will start above the part at the Safety Plane and then ramp into the part along the geoshape, following the contour, until it reaches full cut depth. The cycle will follow the shape back to the start point, overlap by the tool diameter, then ramp out of the shape, again following the contour, back up to the Safety Plane.

Heli-Lead Center is a center cutting (no offset) version of Heli-Lead Outside and Heli-Lead Inside.



Heli-Lead-Center tool path.

Cycle Information		Status Information	
Offset Dim	0.0	Safety Plane	*0.25000
Cut Side	LH	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Round Corners	N	Feedrate/Spindle Speed	
Lead Size	0.0	Feedrate	350.00000
Lead Angle	N	Spindle Speed	18000.00000
Lead Ratio		Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Overlap Amount	AUTO	Calculate	
XY Stock Allowance		Before Codes	
Z Stock Allowance		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Heli-Lead-Center cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (0.0) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the [Offset Dim](#) section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

Note: If setting up a STAYDOWN knowledge, change the default setting to OUTSIDE

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the [Lead Angle](#) section for more information.

Lead Ratio

Lead Ratio determines the angle of the ramp in Z during the lead in and lead out. You can specify the Lead Ratio as a number that reflects the percentage of the angle from its default. That means that if you want a lead that is twice the normal ramp length (shallower angle) enter 2. If you want a lead that is steeper than the default, enter .5.

See the [Lead Ratio](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

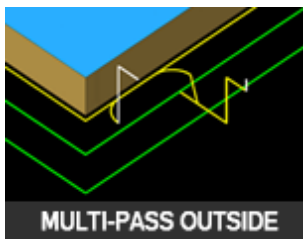
Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.

Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

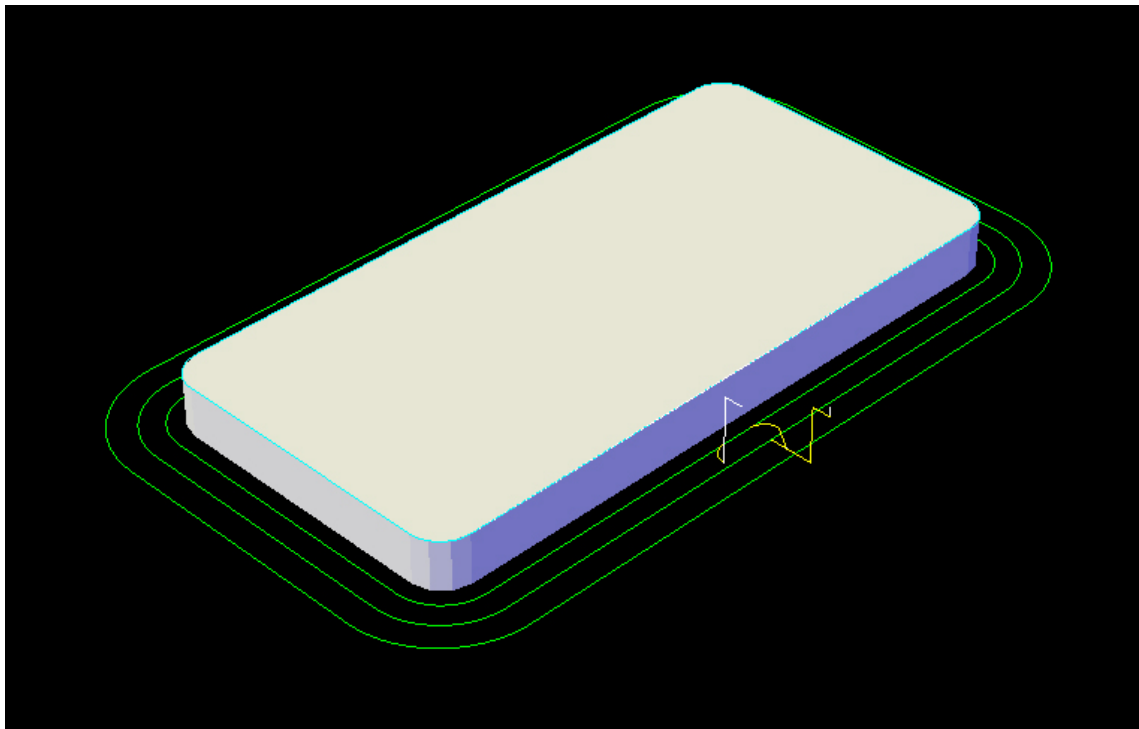
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Multi-Pass-Outside

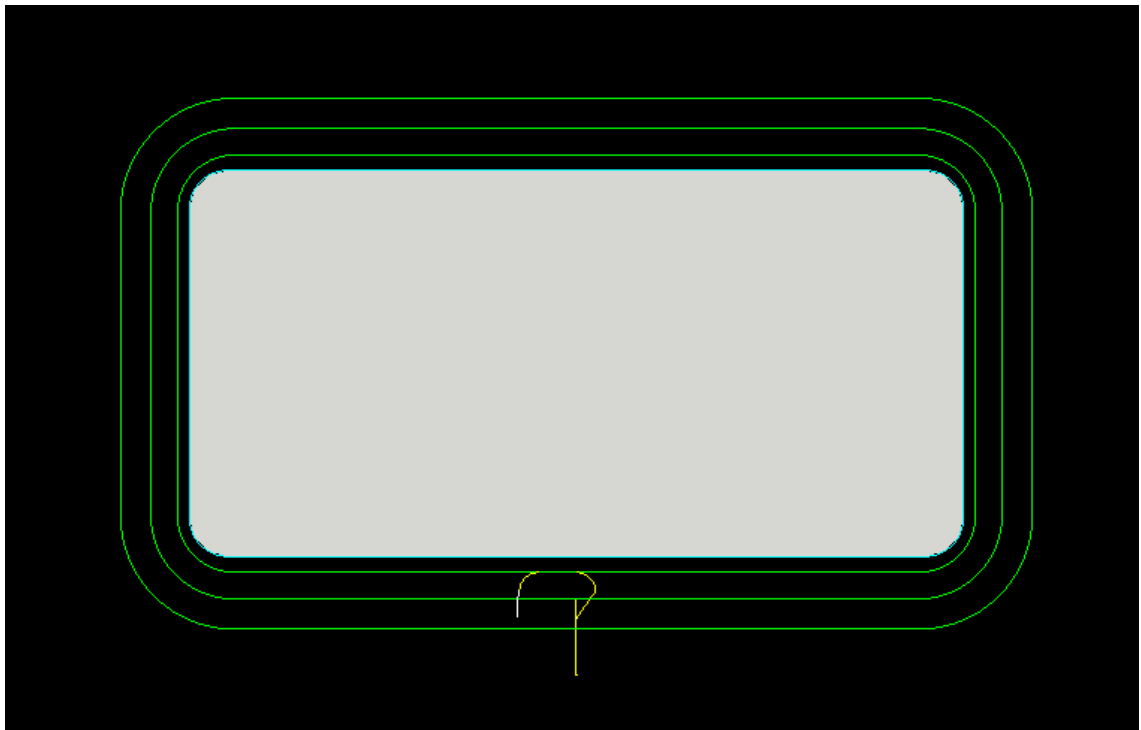


This cycle allows for a rough cut and finish cut on the outside of a shape according to dimensions that you specify in cycle parameters. The cycle creates multiple, offset toolpaths from the outside of the shape towards the finished edge, with specified dimensions for the amount of the first pass, last pass, and cut spacing. You also specify how far away from the finished edge the finish pass is created. The tool will start at the Safety Plane, Plunge to the depth specified in Total Cut Depth or Depth per Pass and then make a straight lead in to the location of the first pass set by the Max Offset. The cutter will make the first pass, then step in towards the finished edge according to the XY Step amount and make another pass until it reaches the offset for the Last Offset. On the Last Offset, the tool will move back and over to a point where it can make a 90° lead-in arc, then cut the finish pass back to the start point, overlap the start point by the tool diameter and make a 90° lead-out arc before retracting back up to the Safety Plane.

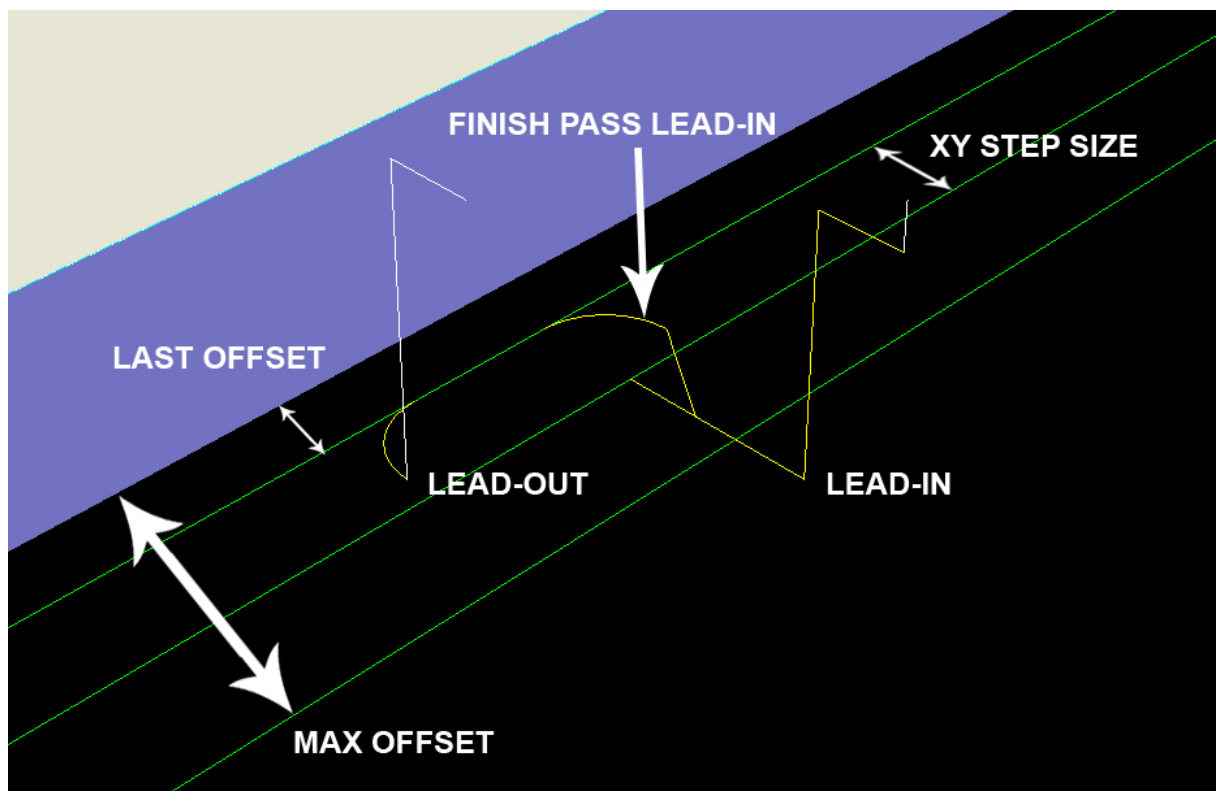
Note: Tabbing is not available with this cycle.



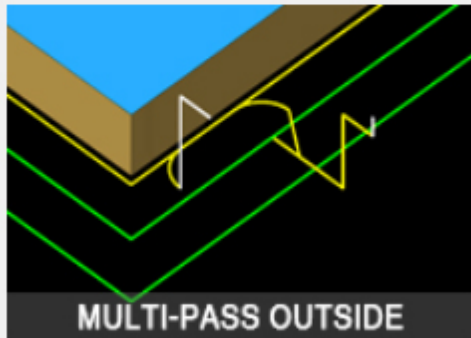
Multi-Pass Outside cut



Multi-Pass Outside cut (from above)



Multi-Pass Outside cut settings

Cycle Information		Status Information	
Cut Direction	CW	Safety Plane	*0.25000
Round Corners	N	Depth Per Pass	1.00000
Lead Size	!*tr*	Total Cut Depth	
Lead Feed		Feedrate/Spindle Speed	
Overlap Amount	AUTO	Feedrate	350.00000
Max Offset		Spindle Speed	18000.00000
XY Step	TW80	Surface FPM	NONE
Last Offset	!*tr*	Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Multi-Pass-Outside cycle parameters

This cycle will only offset shapes that can be offset in AutoCAD, and if the AutoCAD offset command fails, so will this cutting cycle. There is no provision for leaving 'islands' or separate areas that are uncut in the shape. This is not a pocketing cycle. There are some cut properties unique to this cycle, each of them is explained below.

The following parameters effect the toolpath creation:

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any

closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

Max Offset

Max Offset is the distance from the start point to the start of the first pass on the cut. If the cut cycle is Multi-Pass Inside, then the Max Offset will be to the inside of the shape by the specified amount. If the cycle is Multi-Pass Outside, then the Max Offset will be from the start point to the outside of the shape by the Max Offset amount. This is only the distance for the first cut. All subsequent cuts will be determined by either XY Step or Last Offset.

XY Step Size

The XY Step is the distance between each tool path from the first to the last (excluding the finish pass) on the Multi-Pass Inside and Multi-Pass Outside cycles.

You can specify the distance as a numeric value, or you can make a task to calculate a step over amount and place the task name in the XY Step field. An example of a task is TW80, which looks at the tool diameter field and then places an amount equal to 80% of that value in this location.

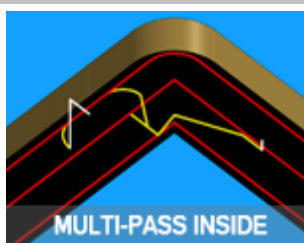
Note: Unless you have created a custom task, TW80 is the only acceptable task call. The only other option would be to use a numeric value.

Last Offset

The Last Offset is the distance from the finish pass to the edge of the shape being cut. This can be any numeric value, but if it exceeds the XY Step value, the cutter will not contact the part on the finish pass. Typically this is set to the radius of the tool in a rough-cut, finish-cut scenario. You can leave some material on the part if you want to clean up the part with a finish cutter.

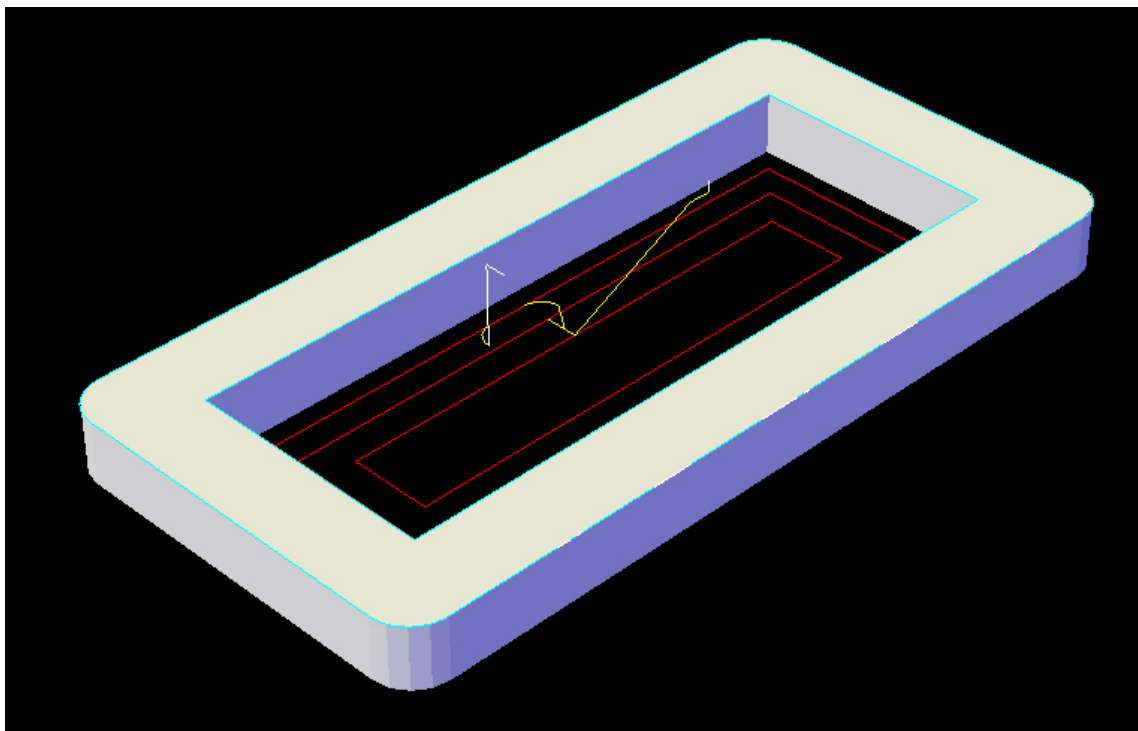
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Multi-Pass-Inside

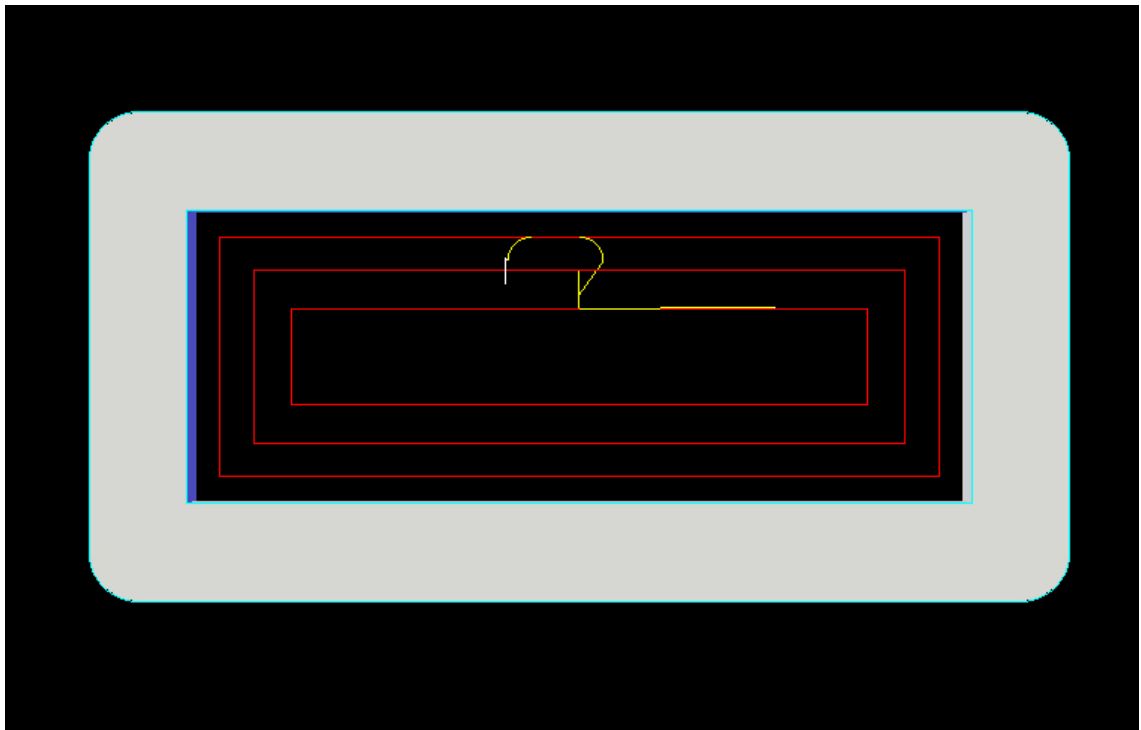


This cycle allows for a rough cut and finish cut on the inside of a shape according to dimensions that you specify in cycle parameters. The cycle creates multiple, offset toolpaths from the inside of the shape towards the finished edge, with specified dimensions for the amount of the first pass, last pass, and cut spacing. You also specify how far away from the finished edge the finish pass is created. The tool will start at the Safety Plane, ramp down to the depth specified in Total Cut Depth or Depth per Pass and then make a straight lead in to the location of the first pass set by the Max Offset. The cutter will make the first pass, then step in towards the finished edge according to the XY Step amount and make another pass until it reaches the offset for the Last Offset. On the Last Offset, the tool will move back and over to a point where it can make a 90° lead-in arc, then cut the finish pass back to the start point, overlap the start point by the tool diameter and make a 90° lead-out arc before retracting back up to the Safety Plane.

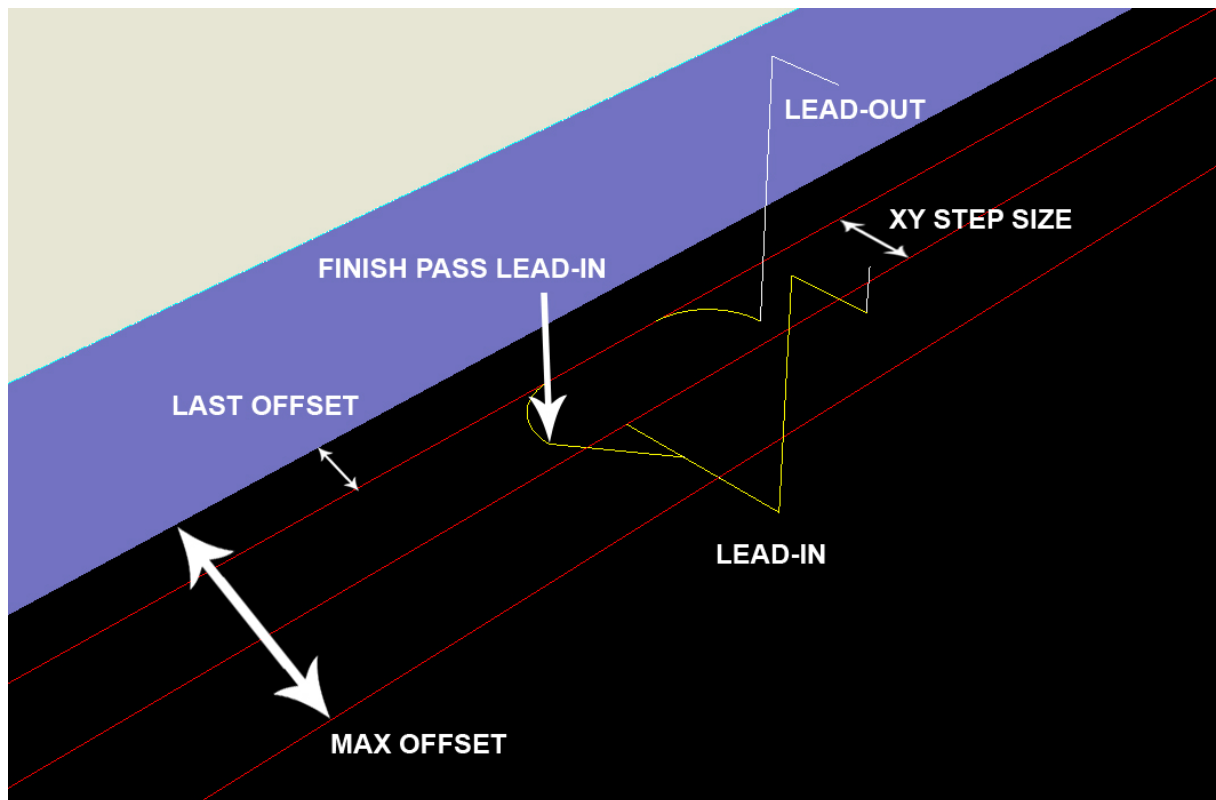
Note: Tabbing is not available with this cycle.



Multi-Pass Inside Cut

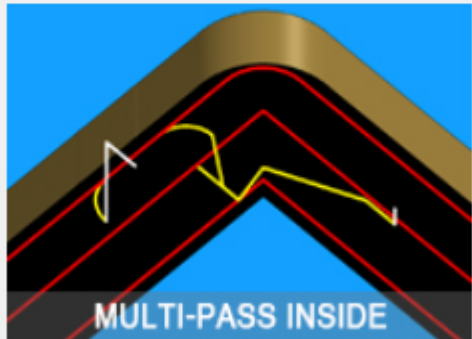


Multi-Pass Inside Cut Example 2



Multi-Pass Inside Cut Settings

One of the main differences between the Multi-Pass Inside and Multi-Pass Outside is that the Inside cycle will make a contour following ramp-in for the first pass. This means that this cycle can be used to remove material from an area inside a shape without a rougher tool having to remove any material first.

Cycle Information		Status Information	
Cut Direction	CCW	Safety Plane	*0.25000
Round Corners	N	Depth Per Pass	1.00000
Lead Size	!*tr*	Total Cut Depth	
Lead Feed		Feederate/Spindle Speed	
Overlap Amount	AUTO	Feederate	350.00000
Max Offset		Spindle Speed	18000.00000
XY Step	TW80	Surface FPM	NONE
Last Offset	!*tr*	Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Multi-Pass Inside cycle parameters

This cycle will only offset shapes that can be offset in AutoCAD, and if the AutoCAD offset command fails, so will this cutting cycle. There is no provision for leaving 'islands' or separate areas that are uncut in the shape. This is not a pocketing cycle. There are some cut properties unique to this cycle, each of them is explained below.

The following parameters effect the toolpath creation:

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

Max Offset

Max Offset is the distance from the start point to the start of the first pass on the cut. If the cut cycle is Multi-Pass Inside, then the Max Offset will be to the inside of the shape by the specified amount. If the cycle is Multi-Pass Outside, then the Max Offset will be from the start point to the outside of the shape by the Max Offset amount. This is only the distance for the first cut. All subsequent cuts will be determined by either XY Step or Last Offset.

XY Step Size

The XY Step is the distance between each tool path from the first to the last (excluding the finish pass) on the Multi-Pass Inside and Multi-Pass Outside cycles.

You can specify the distance as a numeric value, or you can make a task to calculate a step over amount and place the task name in the XY Step field. An example of a task is TW80, which looks at the tool diameter field and then places an amount equal to 80% of that value in this location.

Note: Unless you have created a custom task, TW80 is the only acceptable task call. The only other option would be to use a numeric value.

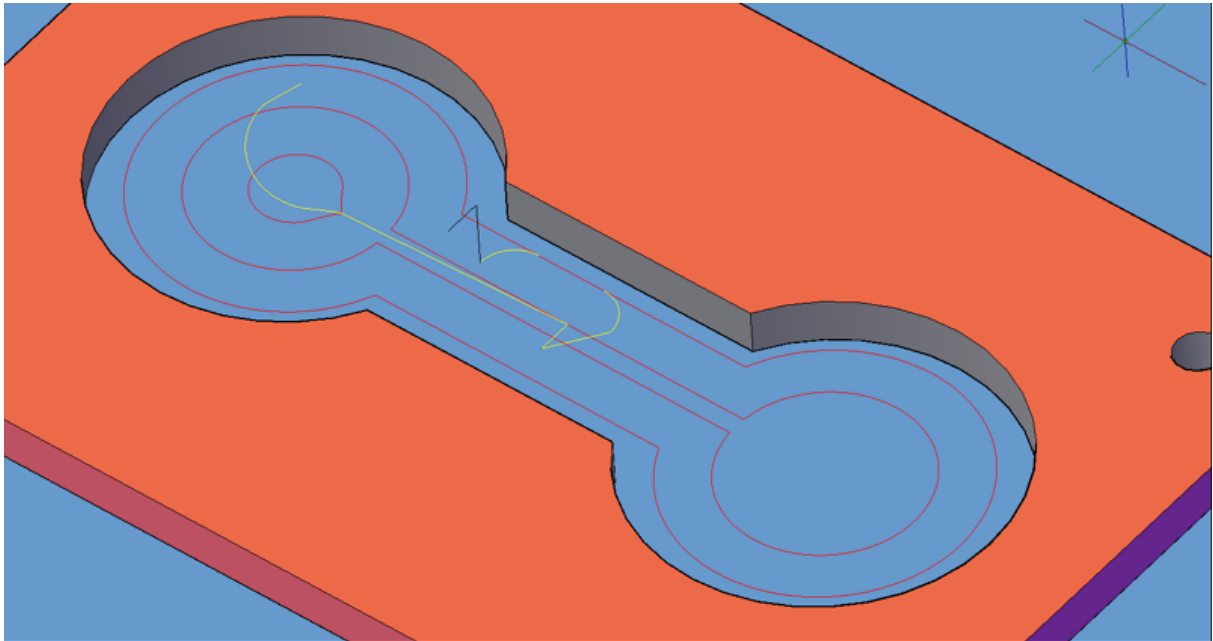
Last Offset

The Last Offset is the distance from the finish pass to the edge of the shape being cut. This can be any numeric value, but if it exceeds the XY Step value, the cutter will not contact the part on the finish pass. Typically this is set to the radius of the tool in a rough-cut, finish-cut scenario. You can leave some material on the part if you want to clean up the part with a finish cutter.

Additional notes for Multi-Pass Inside:

If you are using this cycle to remove all of the material from the inside of a shape, then there are two things to consider.

First, the shape you are cutting must be a regular shape that the AutoCAD offset command can work on to make regular offsets that will clean out the entire area. If the AutoCAD offset command fails to be able to make enough offsets, then this cut cycle will not make the proper tool paths. Shown below is an example of how an irregular shape could fail the offset. For these types of shapes, the Pocketing cycles are appropriate.



Multi-Pass Inside cycle on irregular shape.

Second, if you have a regular shape (rectangle, circle, etc.) and you must remove all the material inside the shape with the Multi-Pass Inside cycle, you will have to do some calculations for the Max Offset. Since Max Offset is the offset distance from the edge of the finished shape to the start of the first cut, you will have to calculate the distance of that first offset tool path.

There are some simple formulas to use for regular shapes.

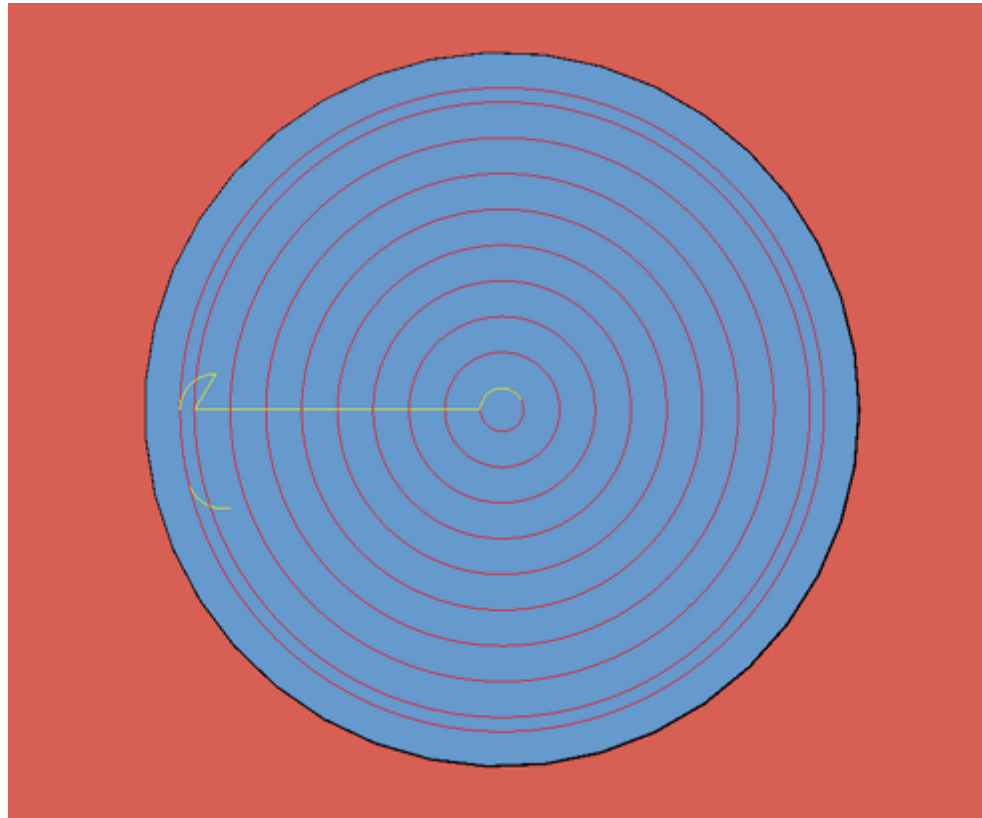
Circular Shapes

To calculate how far to make the Max offset on a circular shape, you can take the Radius of the circle, which would get you to the exact center, then subtract the tool radius to get an offset from the center (otherwise the tool would move down to the center, kind of like a drill with no other motion to make), then add how much of an overlap you want on the first pass.

So, in example to cut a 5" diameter circle, with a .5" Router-Bit, you would start with a Max offset of about 2.35. That is using an overlap of .10 on the first pass.

Diameter / 2 = Circle Radius	$5.0 / 2 = 2.5$
Tool Diameter / 2 = Tool Radius	$0.5 / 2 = .25$
Overlap = .10	$2.5 - .25 = 2.25$
Circle Radius - Tool Radius + Overlap = Max Offset	$2.25 + .10 = \mathbf{2.35}$

The result of using the Max Offset of 2.15, an XY Step of .25 and a Last Offset of .25 would look like this:



Multi-Pass Inside on circular shape.

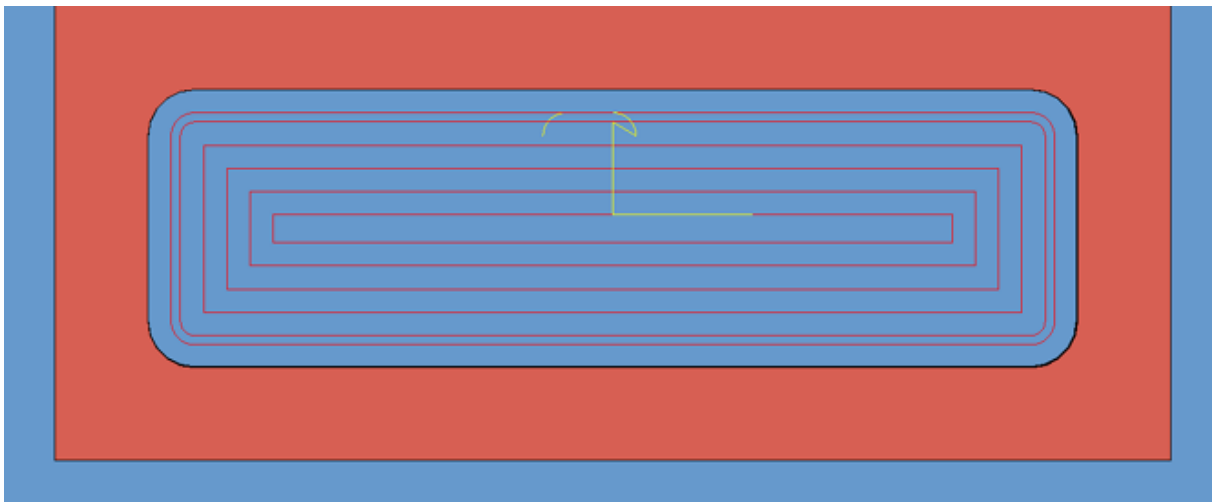
Rectangular Shapes

To calculate the distance of the Max Offset on a rectangular shape, you can take the width of the narrow side divided by 2 (to get to the middle of the shape) then subtract the tool radius and add the overlap amount you want on the first pass.

So, in example, to cut a 10" x 3" rectangular shape, you would need a Max Offset of about 1.35, using an overlap of .10 on the first pass.

Width of narrowest side / 2	$3.0 / 2 = 1.5$
Tool Diameter / 2	$0.5 / 2 = .25$
Overlap on first pass = .10	$1.5 - .25 = 1.25$
Width / 2 - Tool Radius + Overlap = Max Offset	$1.25 + .10 = \mathbf{1.35}$

The result of using the Max Offset of 1.35, an XY Step of .25 and a Last Offset of .25 would look like this:

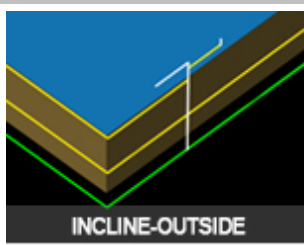


Multi-Pass Inside on a rectangular shape.

The reason for using a smaller first offset amount than the XY Step size or Last offset is only because you need to remove the overlap on the starting pass, each step after that will remove the overlap from the pass before it, so a smaller amount was used. Feel free to use any amount that is just short of the tool radius, as some tools are slightly undersized even when new, and worse as they wear.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

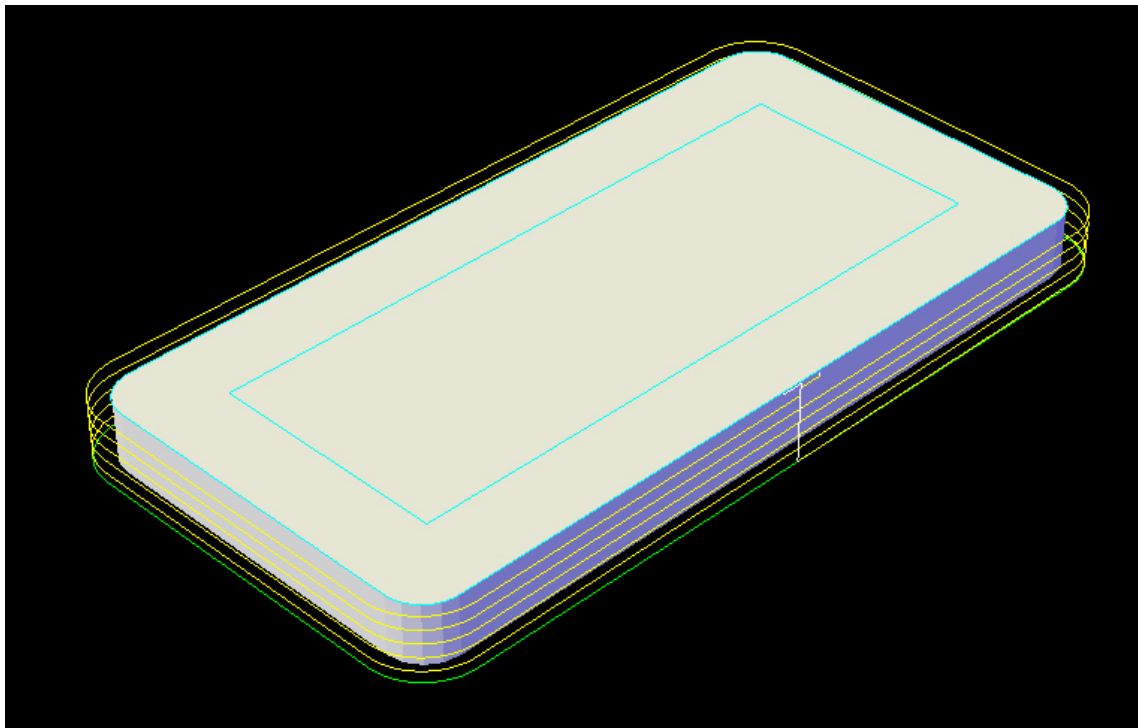
Incline-Outside



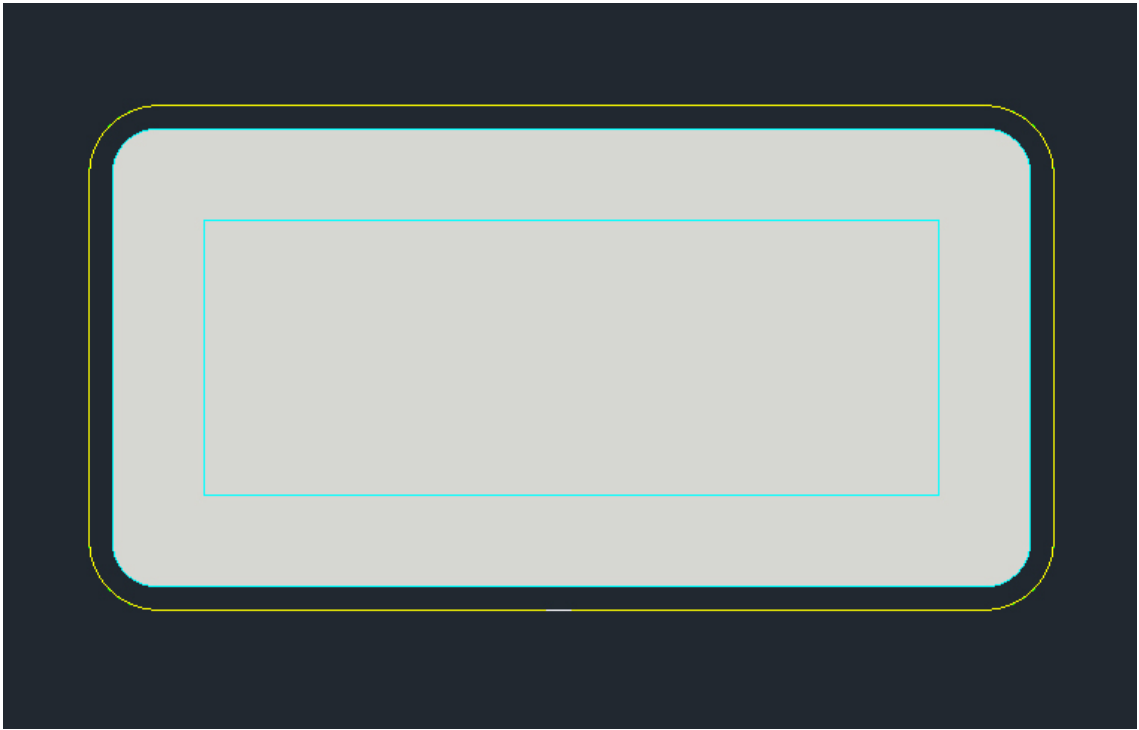
Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.

The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

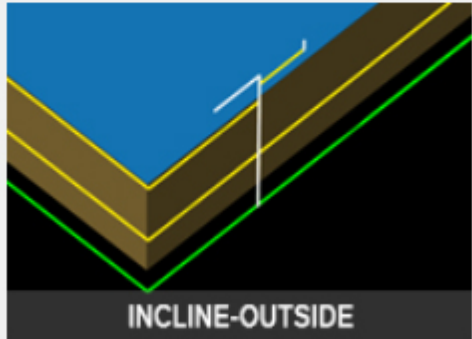
Incline-Outside will default to cutting on the outside of a closed shape.



Incline-Outside cut cycle.



Incline-Outside cut cycle.

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▾	Safety Plane	*0.25000
Cut Side	OUTSIDE ▾	Depth Per Pass	1.00000
Cut Direction	INCLCW ▾	Total Cut Depth	
Round Corners	MDCMP ▾	Feedrate/Spindle Speed	
Lead In	N ▾	Feedrate	350.00000
Lead Out	N ▾	Spindle Speed	18000.00000
Lead Size	LEADSCL ▾	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
XY Stock Allowance		Calculate	
Z Stock Allowance		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Incline-Outside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Depth Per Pass

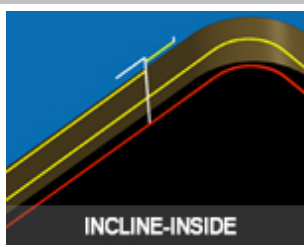
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

Since this cycle is a constant ramp, for the entire length of the cut, there is a little more involved with the depth per pass. If the tool is starting from a point above the part (if tool tip distance to zero is set to any value above 0, which it normally is) then you should take that amount and add it to the total depth and then divide that number by the depth per pass. If you get an even number (no decimal) then that will be the number of passes, otherwise, round up to the next number of passes and then divide the total depth by that number. The result is the depth per pass for each level of the ramp.

For instance, if the tool tip distance to zero is .1 (in the tool parameters) and the total depth is -1.0, then the total movement of the tool is 1.1. If the depth per pass is set to .25, then $1.1 \div .25 = 4.4$. We have to round that number up to 5 otherwise using 4 would give us $1.1 \div 4$ which is .275 and that is more than the depth per pass. So rounding up to 5 gives us $1.1 \div 5$ which is .22 and that is less than the .25 depth per pass, and so you will see 5 levels to the total cut at a spacing of .22 before the final finish pass.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

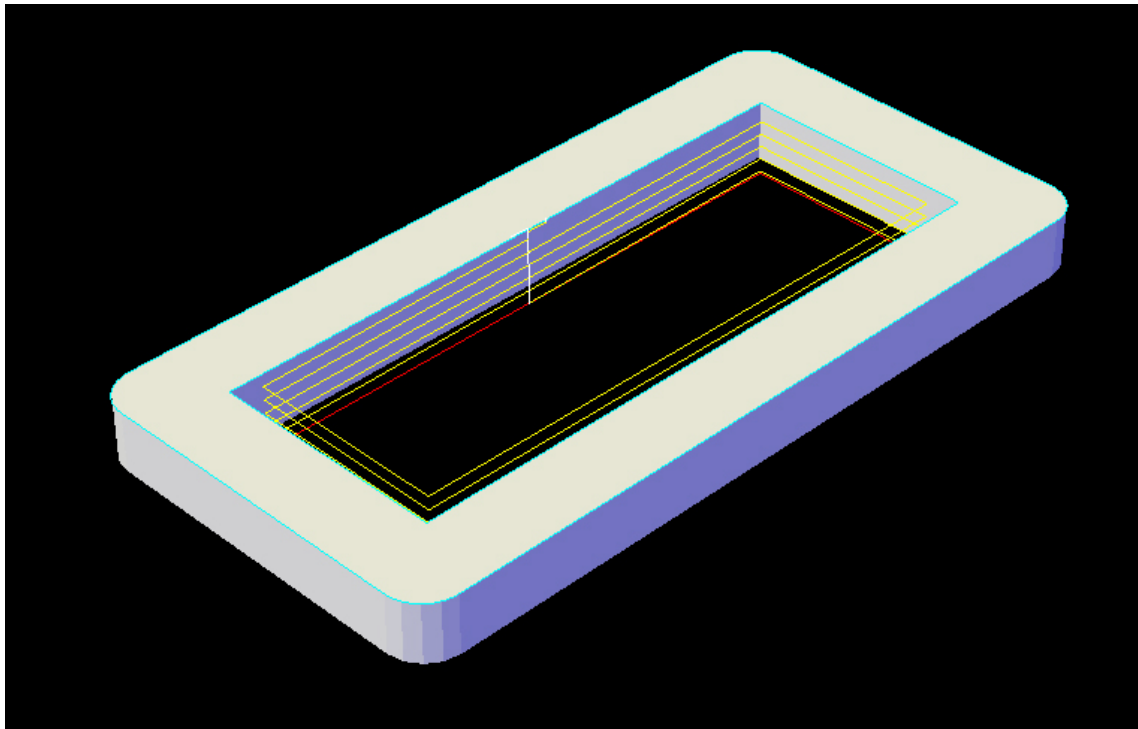
Incline-Inside



Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.

The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

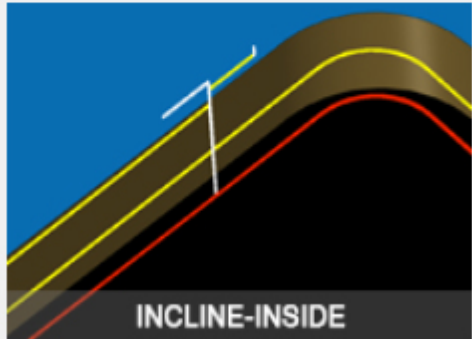
Incline-Inside will default to cutting on the inside of a closed shape.



Incline-Inside cut cycle.



Incline-Inside cut cycle.

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▾	Safety Plane	*0.25000
Cut Side	INSIDE ▾	Depth Per Pass	1.00000
Cut Direction	INCLCCW ▾	Total Cut Depth	
Round Corners	MDCMP ▾	Feedrate/Spindle Speed	
Lead In	N ▾	Feedrate	350.00000
Lead Out	N ▾	Spindle Speed	18000.00000
Lead Size	LEADSCL ▾	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
XY Stock Allowance		Calculate	
Z Stock Allowance		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Incline-Inside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.

Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

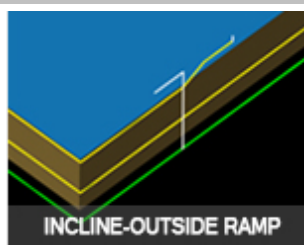
Since this cycle is a constant ramp, for the entire length of the cut, there is a little more involved with the depth per pass. If the tool is starting from a point above the part (if tool tip distance to zero is set to any value above 0, which it normally is) then you should take that amount and add it to the total depth and then divide that number by the depth per pass. If you get an even number (no decimal) then that will be the number of passes, otherwise, round up to the next number of passes and then divide the total depth by that number. The result is the depth per pass for each level of the ramp.

For instance, if the tool tip distance to zero is .1 (in the tool parameters) and the total depth is -1.0, then the total movement of the tool is 1.1. If the depth per pass is set to .25, then $1.1 \div .25 = 4.4$.

We have to round that number up to 5 otherwise using 4 would give us $1.1 \div 4$ which is .275 and that is more than the depth per pass. So rounding up to 5 gives us $1.1 \div 5$ which is .22 and that is less than the .25 depth per pass, and so you will see 5 levels to the total cut at a spacing of .22 before the final finish pass.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Incline-Outside-Ramp

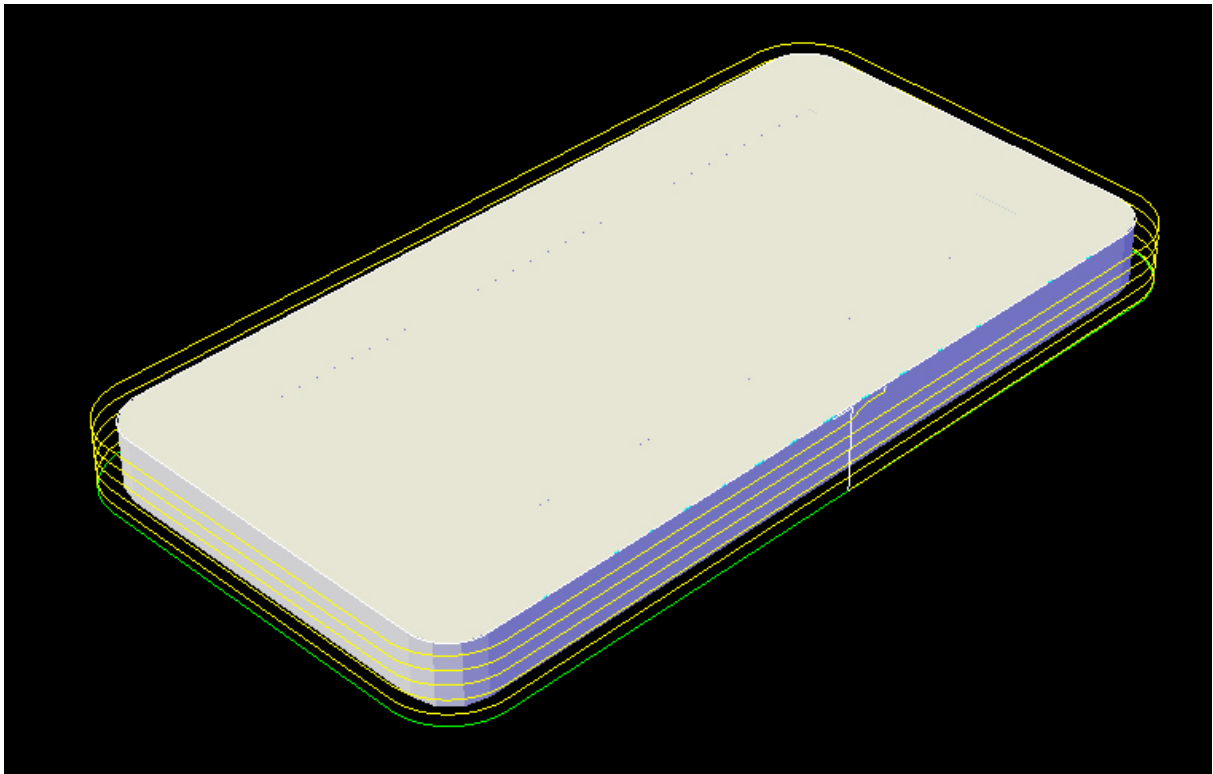


Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.

The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

Incline-Outside will default to cutting on the outside of a closed shape.

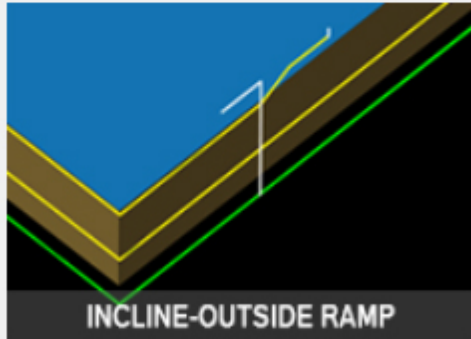
The only difference between Incline-Outside and Incline-Outside Ramp is that the initial move is a ramping lead in to the start of the first pass instead of a plunge move. This is only necessary if the start of the cut will be inside of uncut material. Typically the start of the first cut is above the top of the part. If the start of the first cut cannot be above the top of the part, it may be better for the tool to make a ramp into the material instead of a plunge.



Incline-Outside Ramp Cut Cycle



Incline-Outside Ramp Cut cycle

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC	Safety Plane	*0.25000
Cut Side	OUTSIDE	Depth Per Pass	1.00000
Cut Direction	INCLCW	Total Cut Depth	
Round Corners	MDCMP	Feedrate/Spindle Speed	
Lead In	N	Feedrate	350.00000
Lead Out	N	Spindle Speed	18000.00000
Lead Size	LEADSCL	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
XY Stock Allowance		Calculate	
Z Stock Allowance		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Incline-Outside Ramp Cut Cycle Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Depth Per Pass

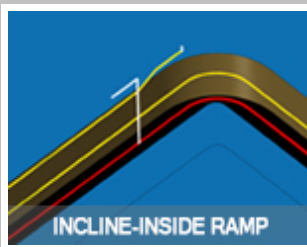
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

Since this cycle is a constant ramp, for the entire length of the cut, there is a little more involved with the depth per pass. If the tool is starting from a point above the part (if tool tip distance to zero is set to any value above 0, which it normally is) then you should take that amount and add it to the total depth and then divide that number by the depth per pass. If you get an even number (no decimal) then that will be the number of passes, otherwise, round up to the next number of passes and then divide the total depth by that number. The result is the depth per pass for each level of the ramp.

For instance, if the tool tip distance to zero is .1 (in the tool parameters) and the total depth is -1.0, then the total movement of the tool is 1.1. If the depth per pass is set to .25, then $1.1 \div .25 = 4.4$. We have to round that number up to 5 otherwise using 4 would give us $1.1 \div 4$ which is .275 and that is more than the depth per pass. So rounding up to 5 gives us $1.1 \div 5$ which is .22 and that is less than the .25 depth per pass, and so you will see 5 levels to the total cut at a spacing of .22 before the final finish pass.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Incline-Inside-Ramp

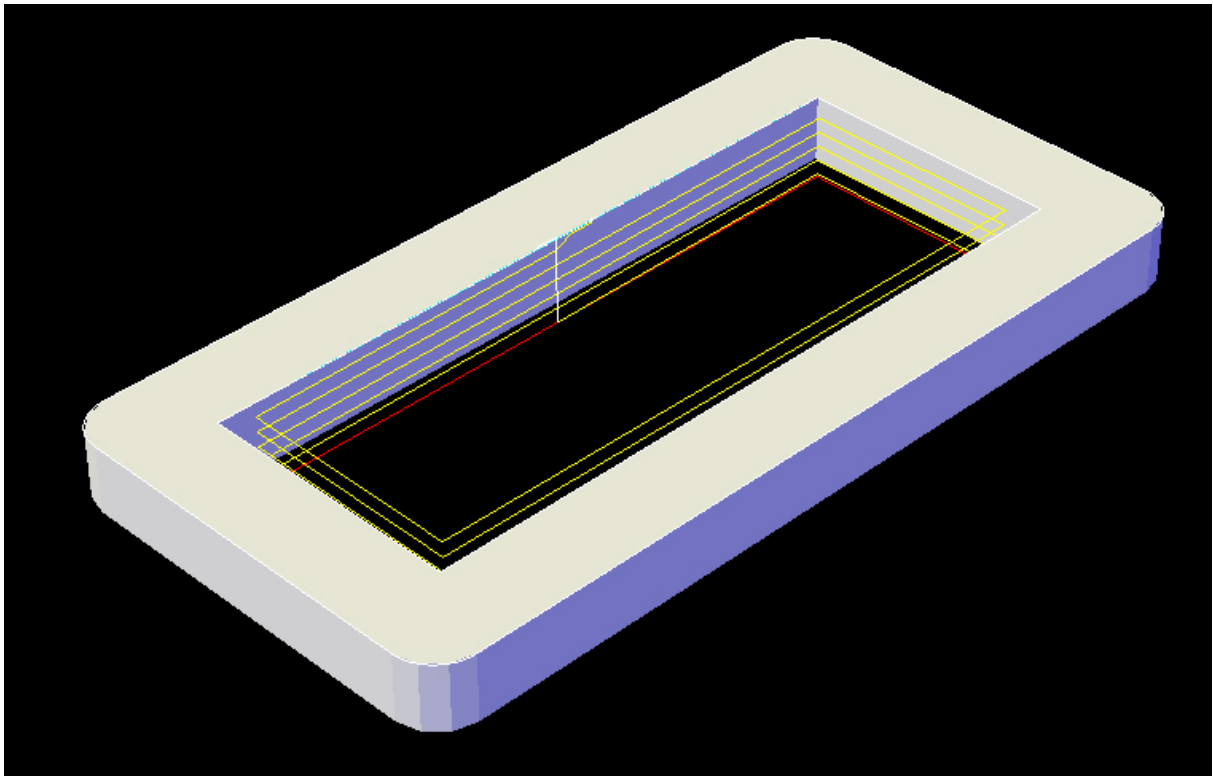


Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.

The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

Incline-Inside Ramp will default to cutting on the inside of a closed shape.

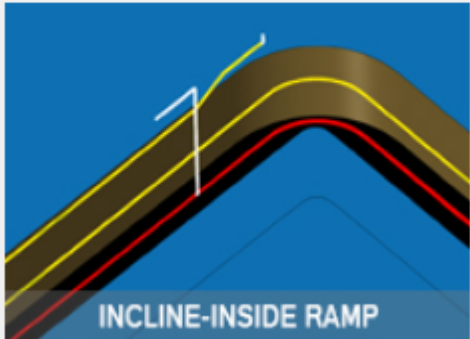
The only difference between Incline-Inside and Incline-Inside Ramp is that the initial move is a ramping lead in to the start of the first pass instead of a plunge move. This is only necessary if the start of the cut will be inside of uncut material. Typically the start of the first cut is above the top of the part. If the start of the first cut cannot be above the top of the part, it may be better for the tool to make a ramp into the material instead of a plunge.



Incline-Inside Ramp Cut Cycle



Incline-Inside Ramp Cut Cycle

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▼	Safety Plane	*0.25000
Cut Side	INSIDE ▼	Depth Per Pass	1.00000
Cut Direction	INCLCCW ▼	Total Cut Depth	
Round Corners	MDCMP ▼	Feedrate/Spindle Speed	
Lead In	N ▼	Feedrate	350.00000
Lead Out	N ▼	Spindle Speed	18000.00000
Lead Size	LEADSCL ▼	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
XY Stock Allowance		Calculate	
Z Stock Allowance		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Incline-Inside-Ramp Cut Cycle Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.

Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Depth Per Pass

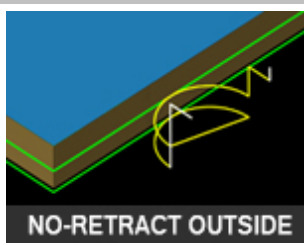
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

Since this cycle is a constant ramp, for the entire length of the cut, there is a little more involved with the depth per pass. If the tool is starting from a point above the part (if tool tip distance to zero is set to any value above 0, which it normally is) then you should take that amount and add it to the total depth and then divide that number by the depth per pass. If you get an even number (no decimal) then that will be the number of passes, otherwise, round up to the next number of passes and then divide the total depth by that number. The result is the depth per pass for each level of the ramp.

For instance, if the tool tip distance to zero is .1 (in the tool parameters) and the total depth is -1.0, then the total movement of the tool is 1.1. If the depth per pass is set to .25, then $1.1 \div .25 = 4.4$. We have to round that number up to 5 otherwise using 4 would give us $1.1 \div 4$ which is .275 and that is more than the depth per pass. So rounding up to 5 gives us $1.1 \div 5$ which is .22 and that is less than the .25 depth per pass, and so you will see 5 levels to the total cut at a spacing of .22 before the final finish pass.

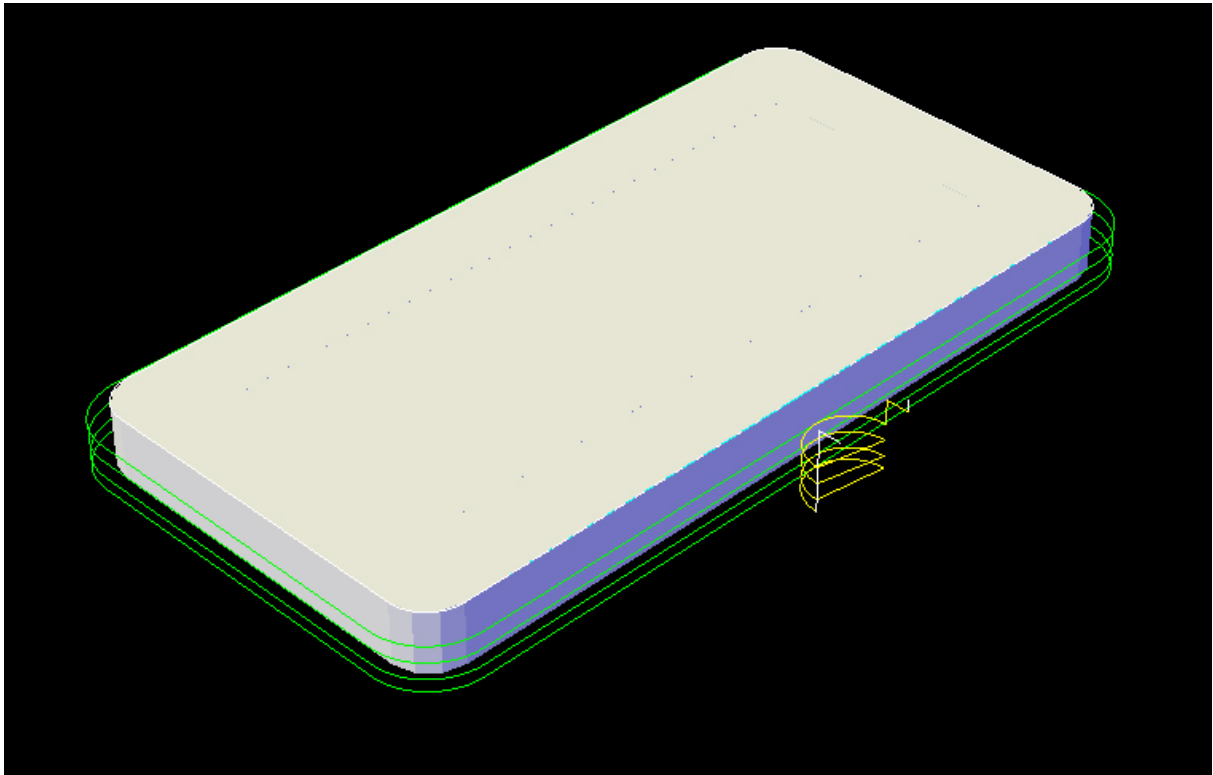
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

No-Retract-Outside

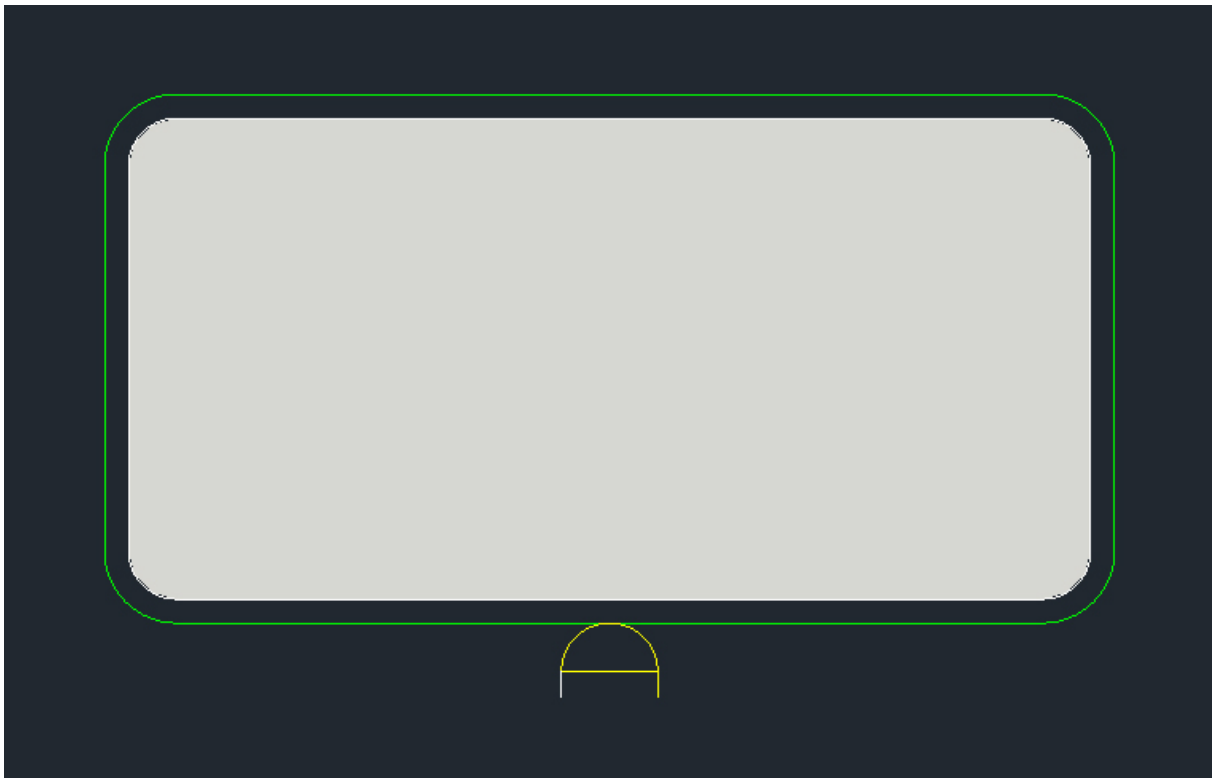


The No-Retract Outside cycle is a profile cutting cycle meant for cuts that need multiple passes in Z. This cycle uses a separate lead-in and lead-out per pass, but instead of retracting the cutter to the Safety Plane between each pass in Z, the tool will move from the end of the lead-out and ramp down to the start of the lead-in on the next pass. Each pass in Z will contain a 90° lead-in and lead-out, and the ramp down between each pass will be performed at the programmed feedrate.

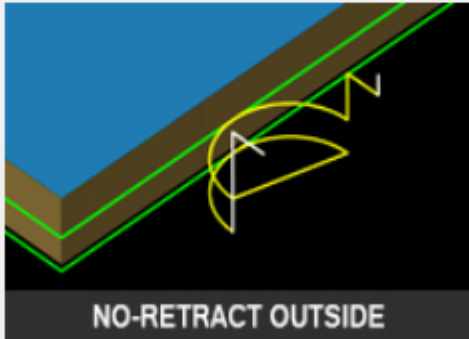
There is no overlap on this cycle by default, as the cutter is constantly overcutting the start point, but an overlap is allowed to be specified in the parameters if desired. This cycle should only be used with multiple depths per pass, as there is no benefit to using it on a single pass in Z.



No-Retract-Outside Cut Cycle



No-Retract-Outside Cut Cycle

Cycle Information		Status Information	
Offset Dim	OFFSZ	Safety Plane	*0.25000
Cut Side	OUTSIDE	Depth Per Pass	1.00000
Cut Direction	CW	Total Cut Depth	
Round Corners	Y	Feedrate/Spindle Speed	
Lead In	LI	Feedrate	350.00000
Lead Out	LO	Spindle Speed	18000.00000
Lead Size	LEADSCL	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Overlap Amount	AUTO	<input type="button" value="Calculate"/>	
XY Stock Allowance		Before Codes	
Z Stock Allowance		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		<input type="button" value="Reset Cycle Settings to Default"/>	

No-Retract-Outside Cut Cycle Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

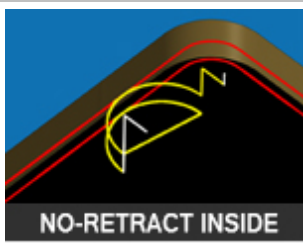
Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.

Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

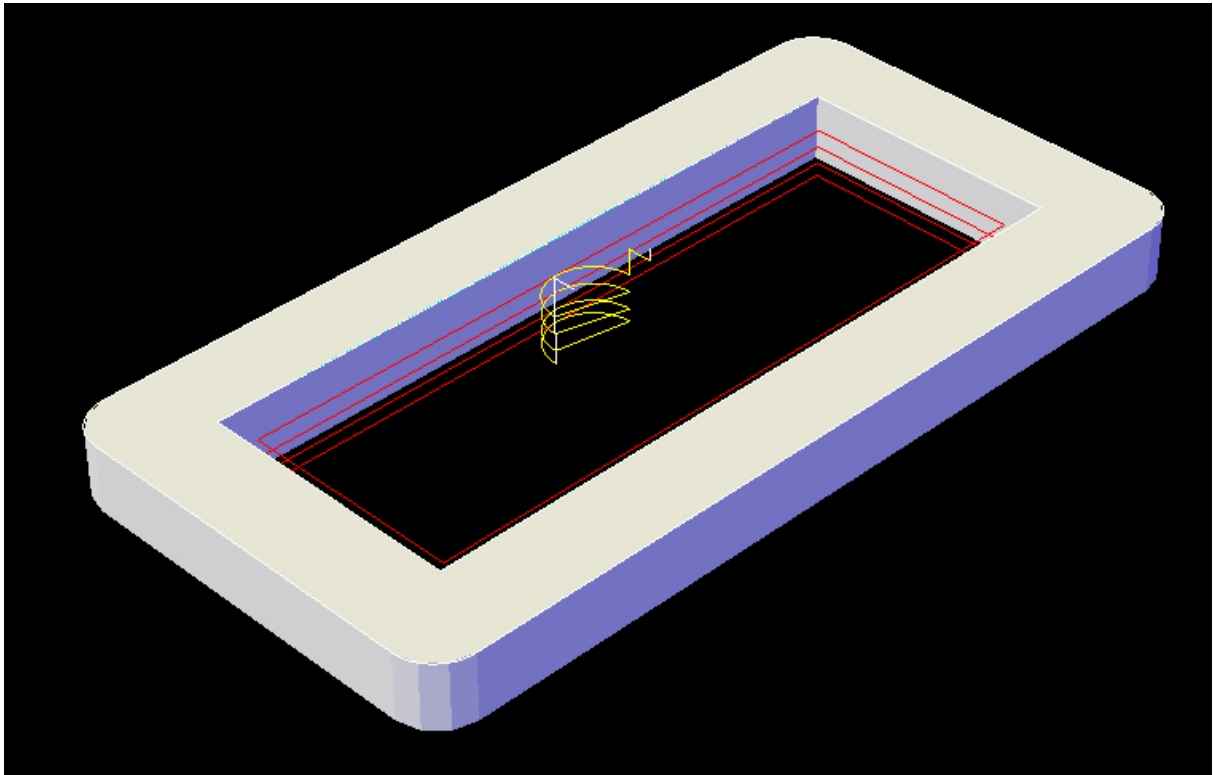
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

No-Retract-Inside

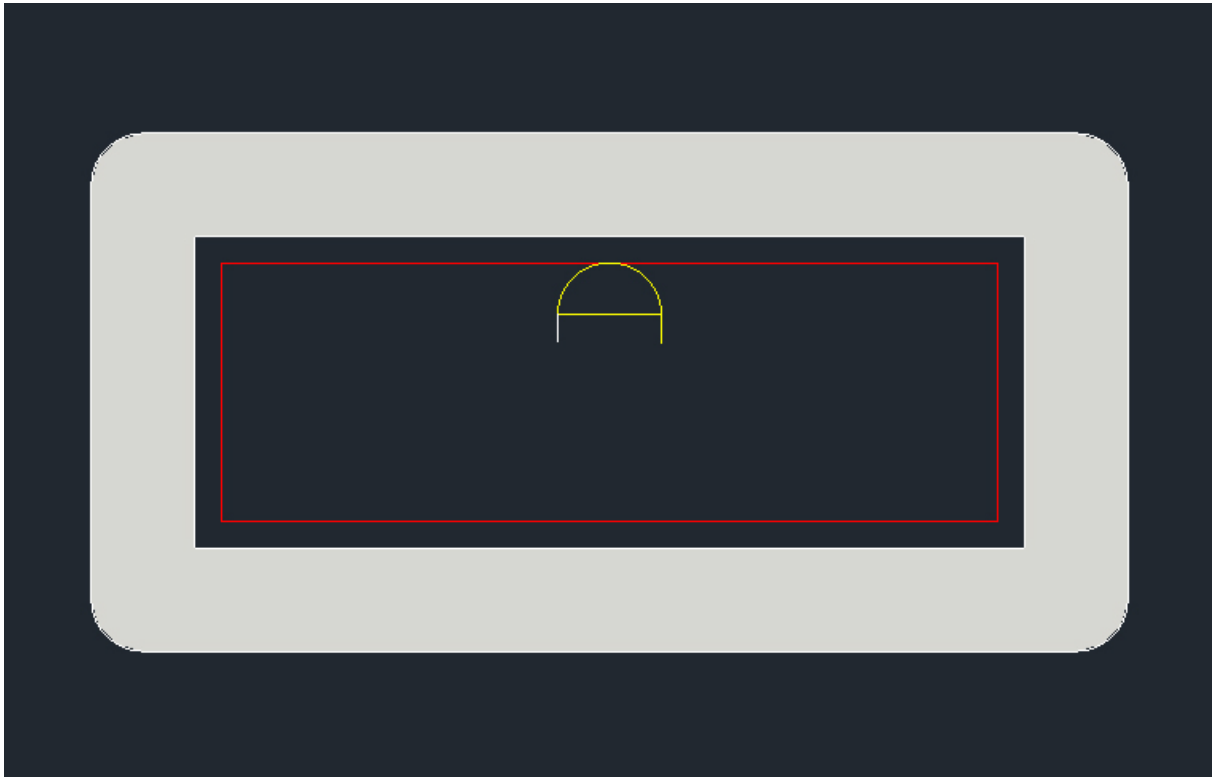


The No-Retract Inside cycle is a profile cutting cycle meant for cuts that need multiple passes in Z. This cycle uses a separate lead-in and lead-out per pass, but instead of retracting the cutter to the Safety Plane between each pass in Z, the tool will move from the end of the lead-out and ramp down to the start of the lead-in on the next pass. Each pass in Z will contain a 90° lead-in and lead-out, and the ramp down between each pass will be performed at the programmed feedrate.

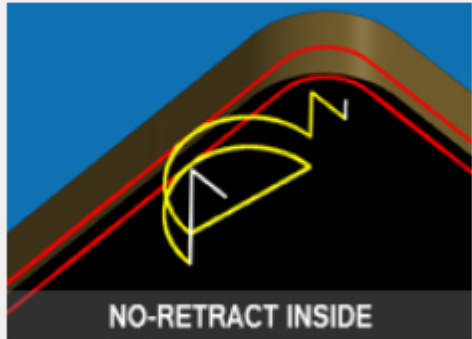
There is no overlap on this cycle by default, as the cutter is constantly overcutting the start point, but an overlap is allowed to be specified in the parameters if desired. This cycle should only be used with multiple depths per pass, as there is no benefit to using it on a single pass in Z.



No-Retract-Inside Cut Cycle



No-Retract-Inside Cut Cycle

Cycle Information		Status Information	
Offset Dim	OFFSZ	Safety Plane	*0.25000
Cut Side	INSIDE	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Round Corners	Y	Feedrate/Spindle Speed	
Lead In	LI	Feedrate	350.00000
Lead Out	LO	Spindle Speed	18000.00000
Lead Size	LEADSCL	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Overlap Amount	AUTO	Calculate	
XY Stock Allowance		Before Codes	
Z Stock Allowance		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

No-Retract-Inside Cut Cycle Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

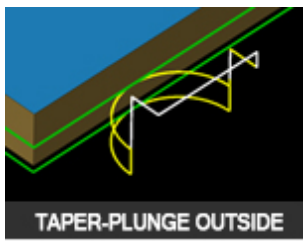
Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.

Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

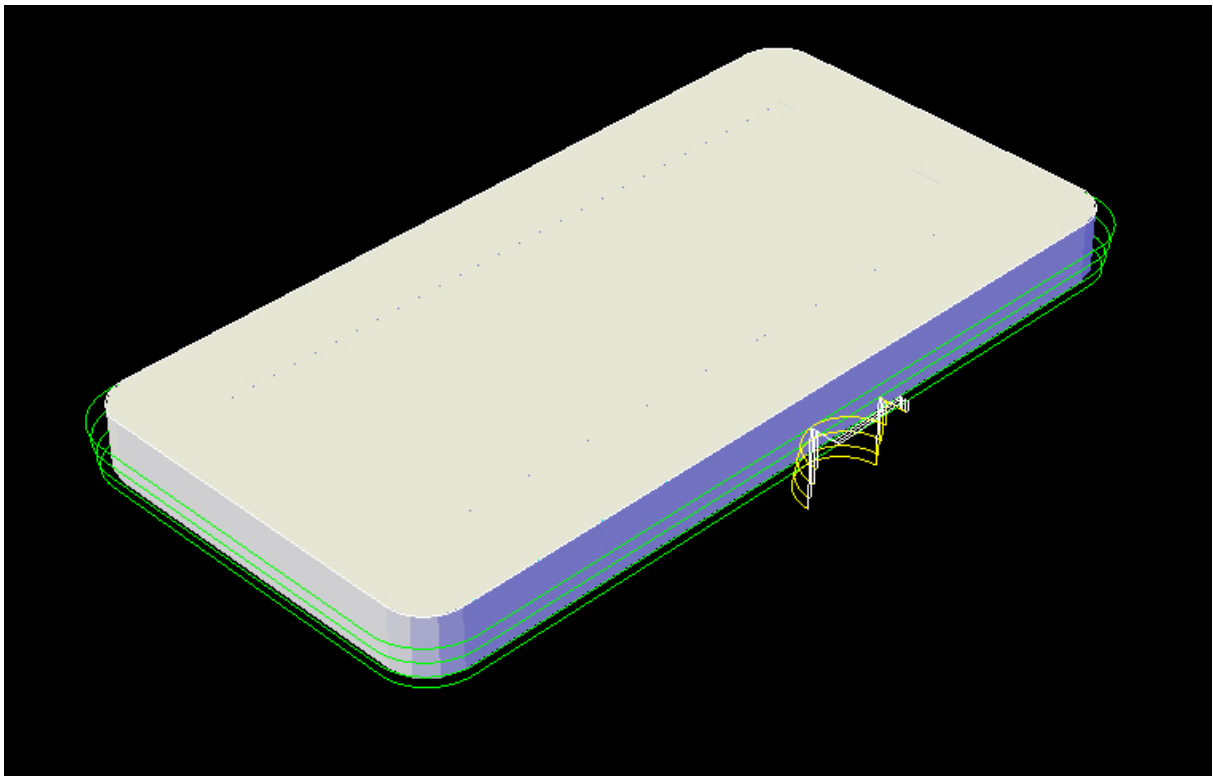
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Taper-Plunge-Outside

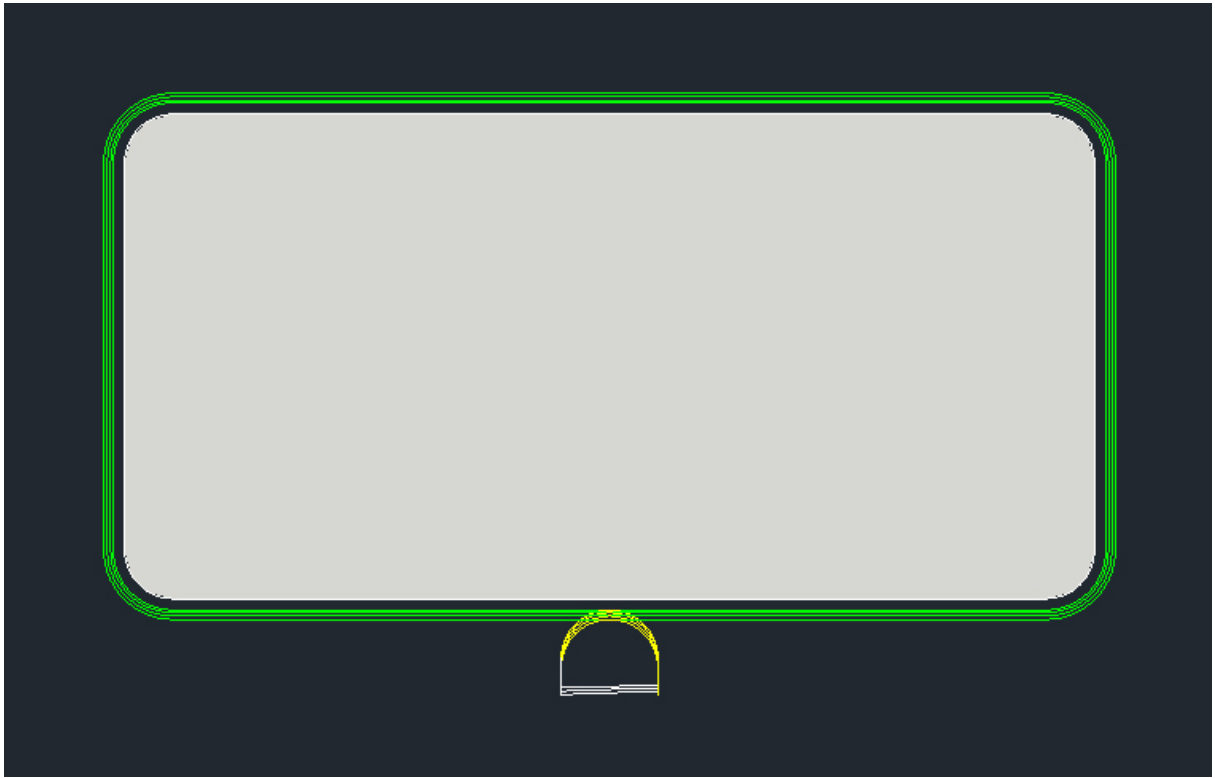


Taper-Plunge cutting is a method of rough and finish cutting with the same tool where the roughing is not only in XY but in Z as well. The cycle will start at the Safety Plane and plunge to the first cut depth and then make a 90° lead-in to make the first pass. The offset of the first pass is typically offset in XY away from the edge of the finished shape. After the first pass takes place, the cutter makes a 90° lead out, retracts the cutter and then moves over to the start of the second cut, which will be closer in XY to the finished edge and lower in Z. It will make the same lead-in move, cut the profile shape and then lead-out and retract, next moving over closer to the finished edge in XY and then plunging in Z down to the next depth. Depending on the set parameters this can occur many times, each time stepping in and down closer to the finished shape of the part, which will be the last pass.

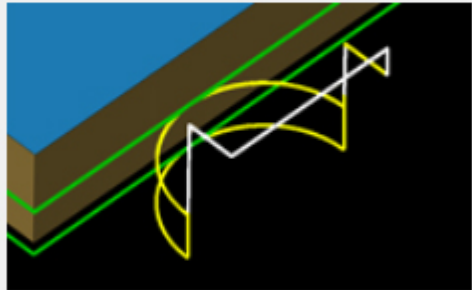
You must use multiple depths per pass to have the cycle parameters create any effect on the tool path.



Taper-Plunge-Outside Cut Cycle



Taper-Plunge-Outside Cut Cycle

Cycle Information		Status Information	
Offset Dim	TPROFF	Safety Plane	*0.25000
Cut Side	OUTSIDE	Depth Per Pass	1.00000
Cut Direction	CW	Total Cut Depth	
Round Corners	MDCMP	Feederate/Spindle Speed	
Lead In	LI	Feederate	350.00000
Lead Out	LO	Spindle Speed	18000.00000
Lead Size	LEADSCL	Surface FPM	NONE
Rough Amount		Units Per Revolution	NONE
Finish Allowance		Calculate	
Taper Angle		Before Codes	
Lead Feed		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
		 <p>TAPER-PLUNGE OUTSIDE</p>	
		Reset Cycle Settings to Default	

Taper-Plunge-Outside Cut Cycle Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit your particular cutting needs.

The value set by default (TPROFF) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Rough Amount

This parameters specifies how far each pass is from one another in the XY plane. Even though the cuts can step in the Z axis, they have an XY step over amount as this would control the amount of material removed by the tool on each pass.

The Z axis step is controlled by the Depth per Pass and Taper Angle.

Finish Allowance

Finish is the offset amount from the edge of the part where the finish pass is placed, in the XY plane.

Typically for a finish cut this would be the radius of the tool, for a roughing cut it would be the tool radius plus the amount of material you desire to leave for the finish cutter to remove.

Taper Angle

The angle in degrees to try and step each cut as they get closer or further away from the part. This would be the angle between the passes in Z. A negative numeric value will start the cut away from the geometry edge and will finish at the 'Finish Allowance' when it reaches Z depth.

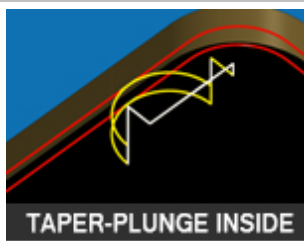
Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

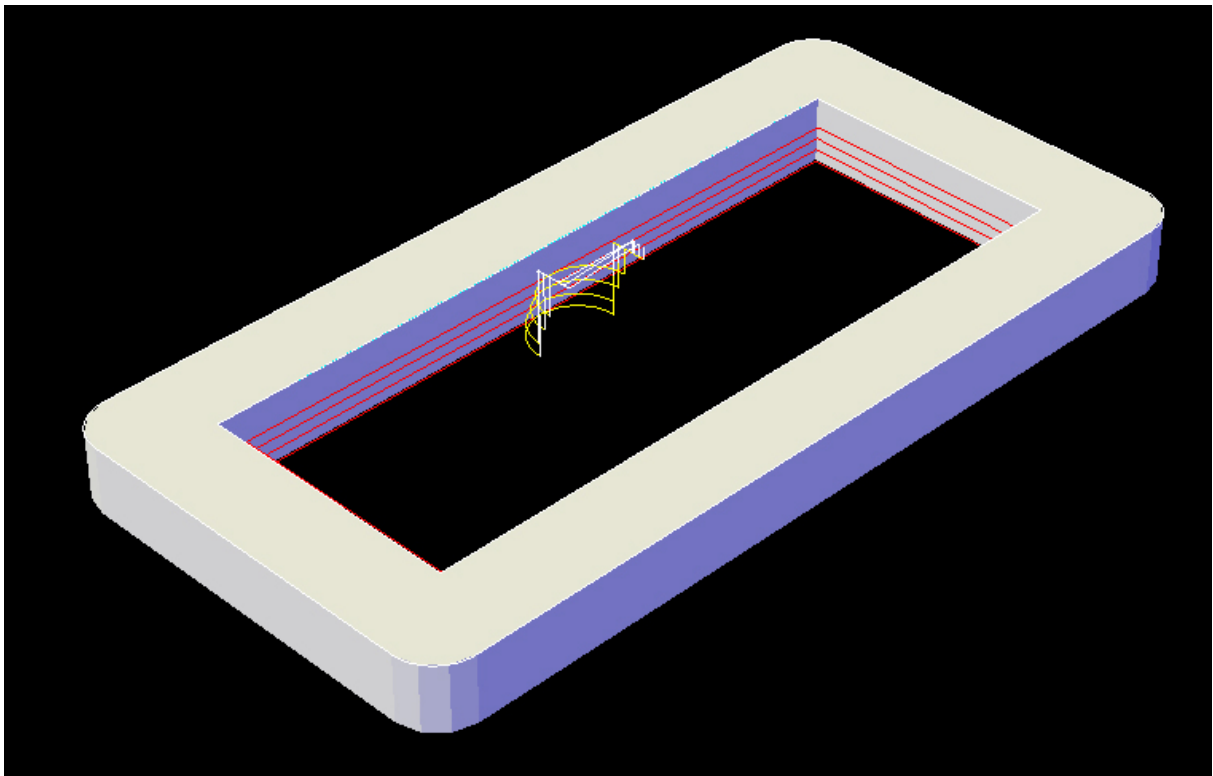
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Taper-Plunge-Inside

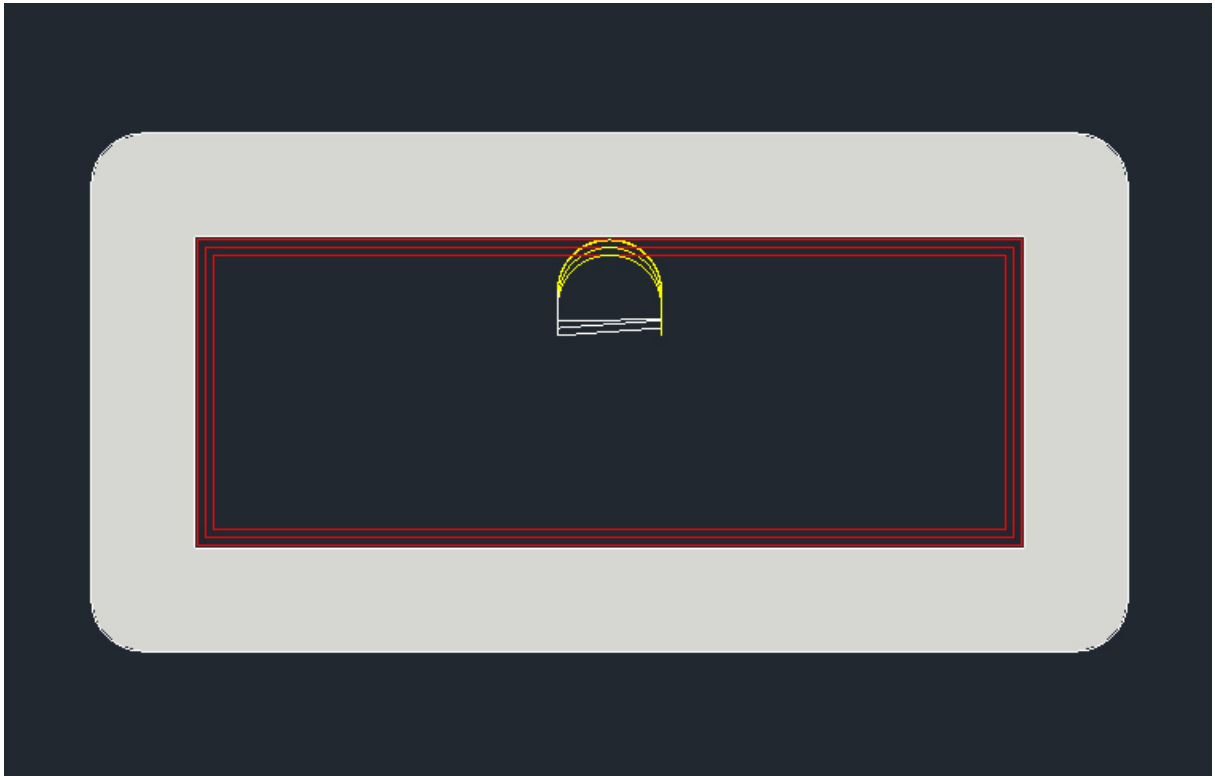


Taper-Plunge cutting is a method of rough and finish cutting with the same tool where the roughing is not only in XY but in Z as well. The cycle will start at the Safety Plane and plunge to the first cut depth and then make a 90° lead-in to make the first pass. The offset of the first pass is typically offset in XY away from the edge of the finished shape. After the first pass takes place, the cutter makes a 90° lead out, retracts the cutter and then moves over to the start of the second cut, which will be closer in XY to the finished edge and lower in Z. It will make the same lead-in move, cut the profile shape and then lead-out and retract, next moving over closer to the finished edge in XY and then plunging in Z down to the next depth. Depending on the set parameters this can occur many times, each time stepping in and down closer to the finished shape of the part, which will be the last pass.

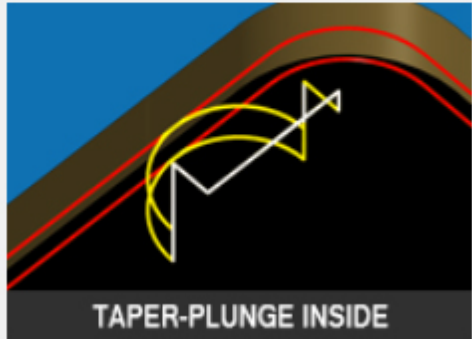
You must use multiple depths per pass to have the cycle parameters create any effect on the tool path.



Taper-Plunge-Inside Cut Cycle



Taper-Plunge-Inside Cut Cycle

Cycle Information		Status Information	
Offset Dim	TPROFF	Safety Plane	*0.25000
Cut Side	INSIDE	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Round Corners	MDCMP	Feedrate/Spindle Speed	
Lead In	LI	Feedrate	350.00000
Lead Out	LO	Spindle Speed	18000.00000
Lead Size	LEADSCL	Surface FPM	NONE
Rough Amount		Units Per Revolution	NONE
Finish Allowance		Calculate	
Taper Angle		Before Codes	
Lead Feed		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Taper-Plunge-Inside Cut Cycle Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit your particular cutting needs.

The value set by default (TPROFF) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Rough Amount

This parameters specifies how far each pass is from one another in the XY plane. Even though the cuts can step in the Z axis, they have an XY step over amount as this would control the amount of material removed by the tool on each pass.

The Z axis step is controlled by the Depth per Pass and Taper Angle.

Finish Allowance

Finish is the offset amount from the edge of the part where the finish pass is placed, in the XY plane.

Typically for a finish cut this would be the radius of the tool, for a roughing cut it would be the tool radius plus the amount of material you desire to leave for the finish cutter to remove.

Taper Angle

The angle in degrees to try and step each cut as they get closer or further away from the part. This would be the angle between the passes in Z. A negative numeric value will start the cut away from the geometry edge and will finish at the 'Finish Allowance' when it reaches Z depth.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Profile-Inside-Auto

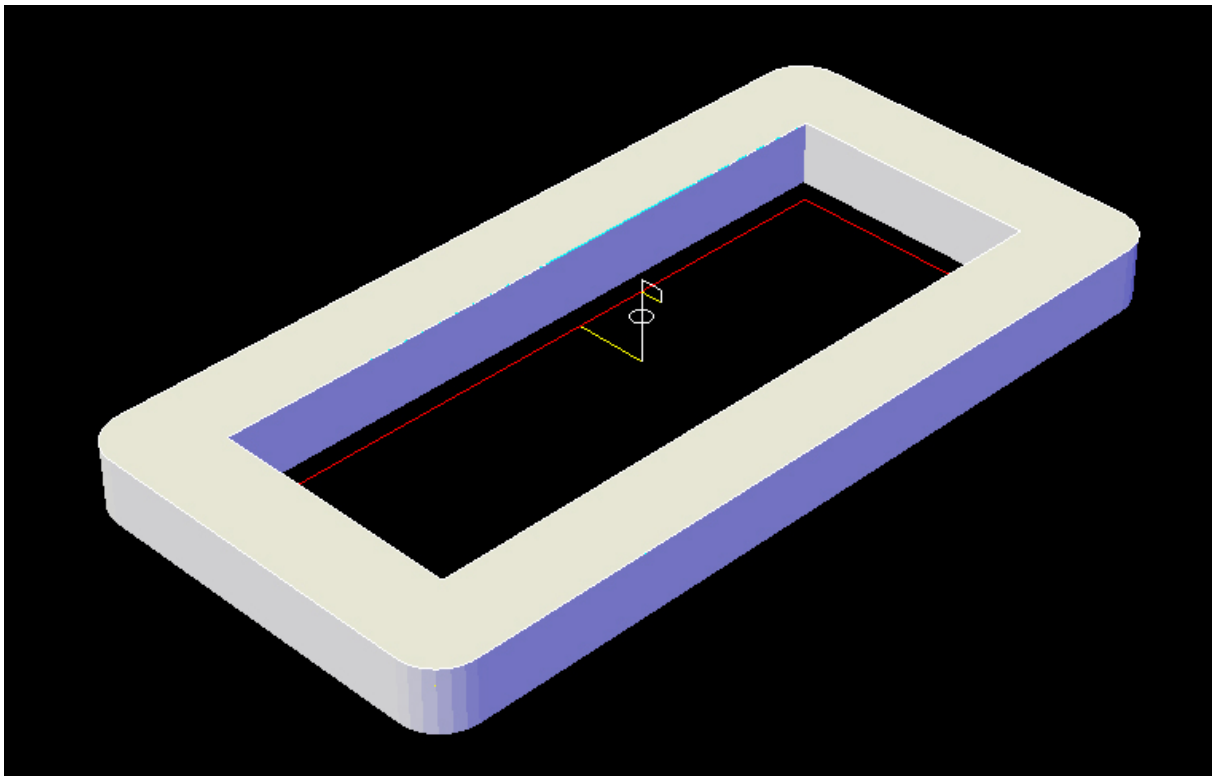


Profile-Inside cycles will start the cut in a hole that has been created for the purpose of getting the tool down to the proper cut depth without plunging the cutter into the material.

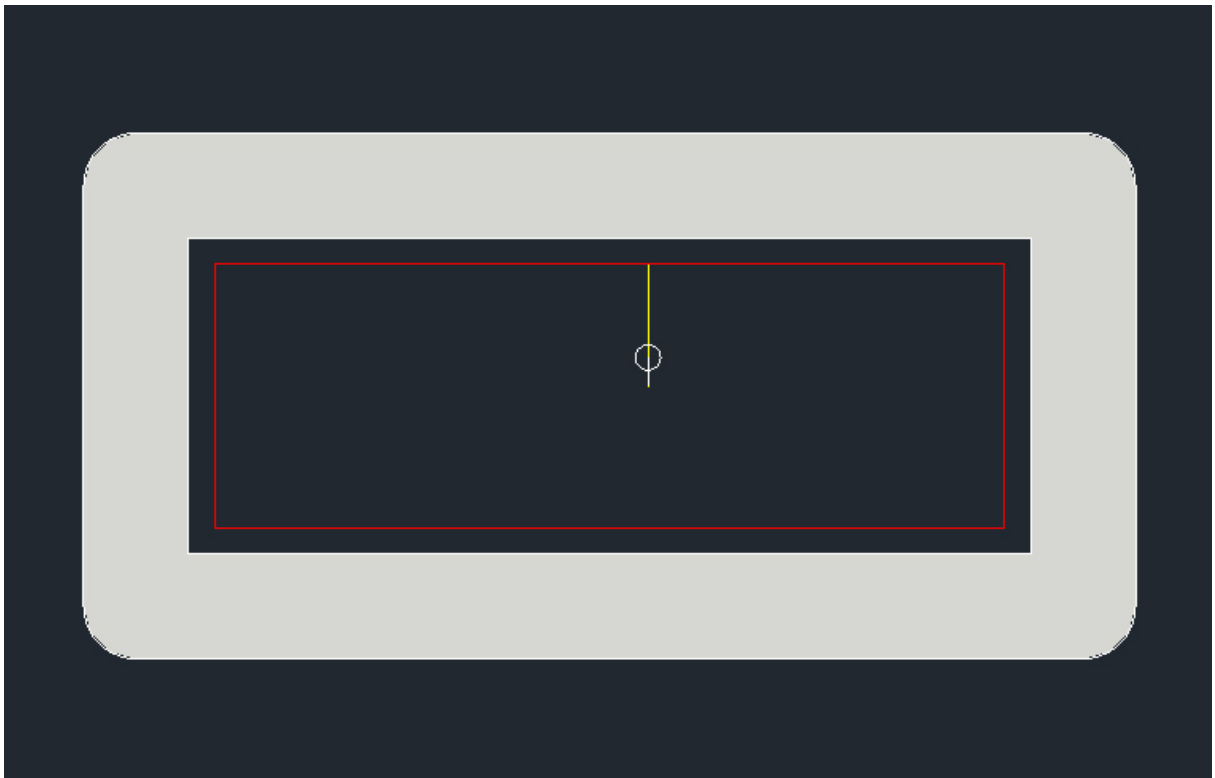
The cycle will start at the Safety Plane, plunge down to the Total Cut Depth inside the hole that has been created for that purpose. Then the cutter will machine a path to the start point of the shape and cut around the inside of the profile back to the start point and then back to the lead hole where it started and at that point it will retract back up to the Safety Plane.

Profile-Inside Auto requires a circle on layer 'CSTART' in order to automatically have a starting hole location. The circle on layer CSTART should be drilled or milled out before this cut cycle occurs, insuring that the cutter has a safe place to plunge.

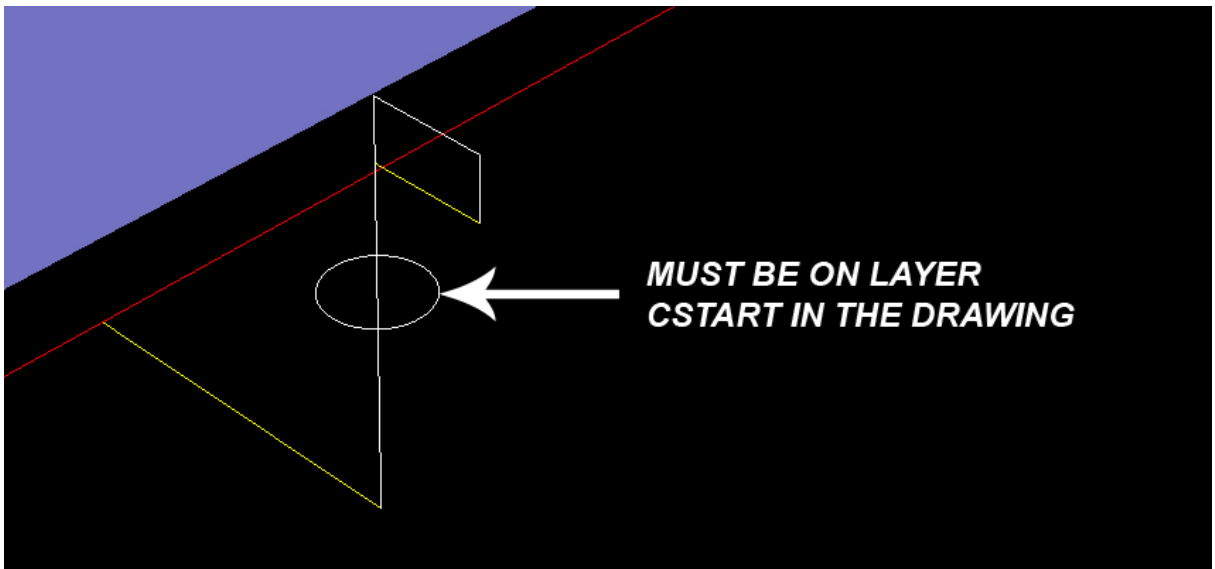
Note: Profile-Inside cycles cannot do multiple depths. Make sure the 'Depth Per Pass' is set sufficient to complete the cut in 1 pass.




Profile-Inside-Automatic Cut Cycle



Profile-Inside-Automatic Cut Cycle



Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▾	Safety Plane	*0.25000
Cut Side	INSIDE ▾	Depth Per Pass	1.00000
Cut Direction	CCW ▾	Total Cut Depth	
Round Corners	N ▾	Feedrate/Spindle Speed	
XY Stock Allowance	▾	Feedrate	350.00000
Z Stock Allowance	▾	Spindle Speed	18000.00000
Lead Feed	▾	Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Profile-Inside-Auto Cut Cycle Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

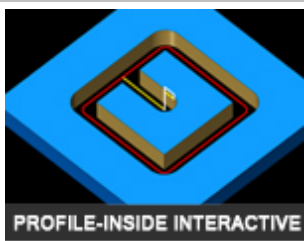
Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Profile-Inside-Interactive

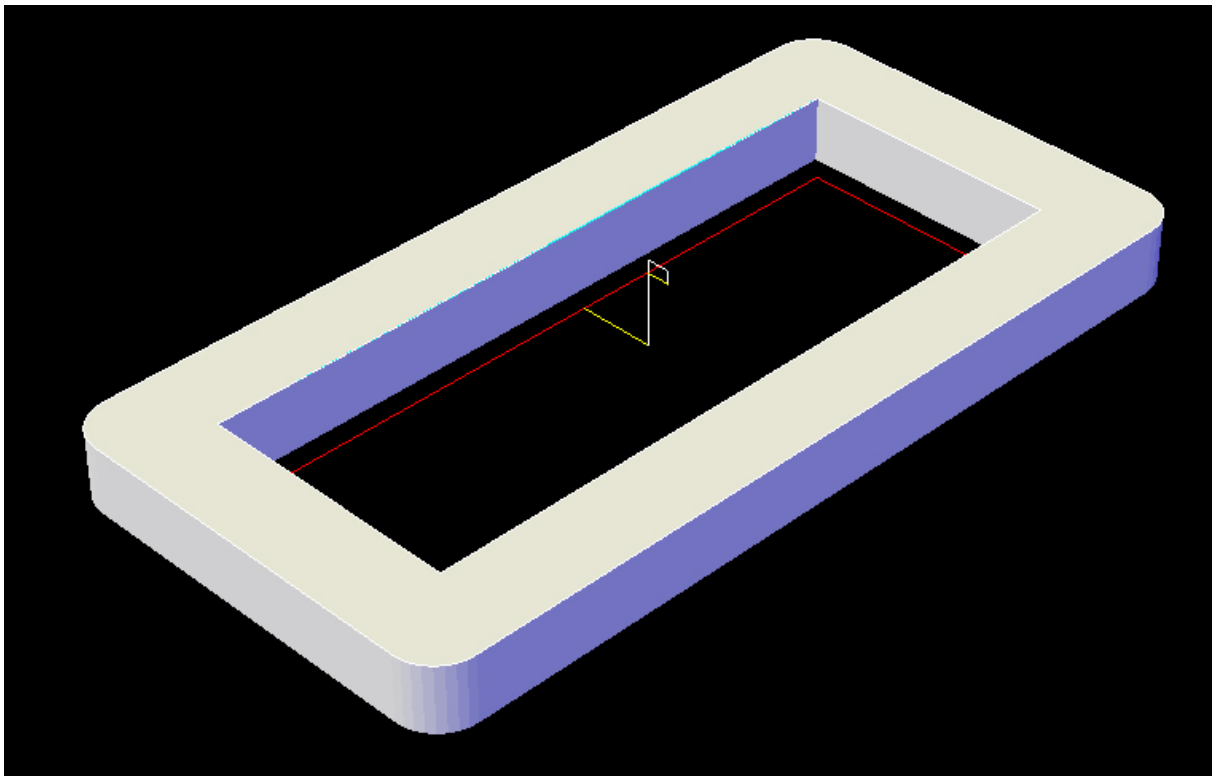


Profile-Inside cycles will start the cut in a hole that has been created for the purpose of getting the tool down to the proper cut depth without plunging the cutter into the material.

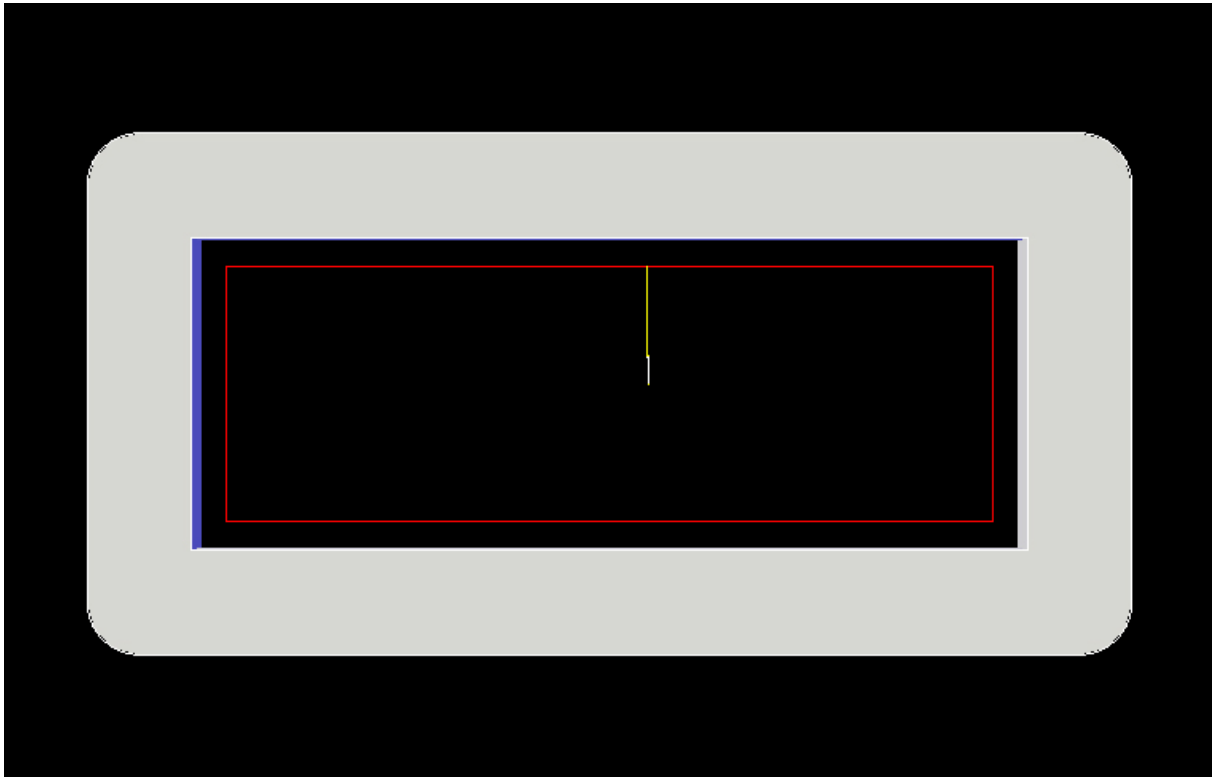
The cycle will start at the Safety Plane, plunge down to the Total Cut Depth inside the hole that has been created for that purpose. Then the cutter will machine a path to the start point of the shape and cut around the inside of the profile back to the start point and then back to the lead hole where it started and at that point it will retract back up to the Safety Plane.

Profile-Inside Interactive requires you to pick the center of a hole to start in during the cut. The circle on should be drilled or milled out before this cut cycle occurs, insuring that the cutter has a safe place to plunge. No circle needs to be in the drawing but you should be sure to select a location that the material will already be removed prior to this cut.


Note: Profile-Inside cycles cannot do multiple depths. Make sure the 'Depth Per Pass' is set sufficient to complete the cut in 1 pass.



Profile-Inside Interactive Cut Cycle.



Profile-Inside Interactive Cut Cycle.

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▼	Safety Plane	*0.25000
Cut Side	INSIDE ▼	Depth Per Pass	1.00000
Cut Direction	CCW ▼	Total Cut Depth	
Round Corners	N ▼	Feedrate/Spindle Speed	
XY Stock Allowance		Feedrate	350.00000
Z Stock Allowance		Spindle Speed	18000.00000
Lead Feed		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Profile-Inside-Interactive Cut Cycle Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Drilling and Hole Cutting Cycles

There are various drilling and hole cutting cycles inside Router-CIM. These cycles are designed with specific purposes and are all explained in as much detail as possible to ensure you can select the ideal cycle for the task at hand.

There are different cycles for drilling than there are for hole cutting as there are instances where you must machine a hole with a router-bit or end mill instead of drilling with a drill-bit. These cycles are meant for the instances where the job is performed with a tool suitable to remove all the material in the hole and not leave a slug which could be ejected from the material and cause injury.

If it is not possible to cut the hole with a large enough tool to remove the slug, sometimes it is better to pocket cut the slug first and then finish cut the inside of the hole to obtain the best finish, and the safest practice.

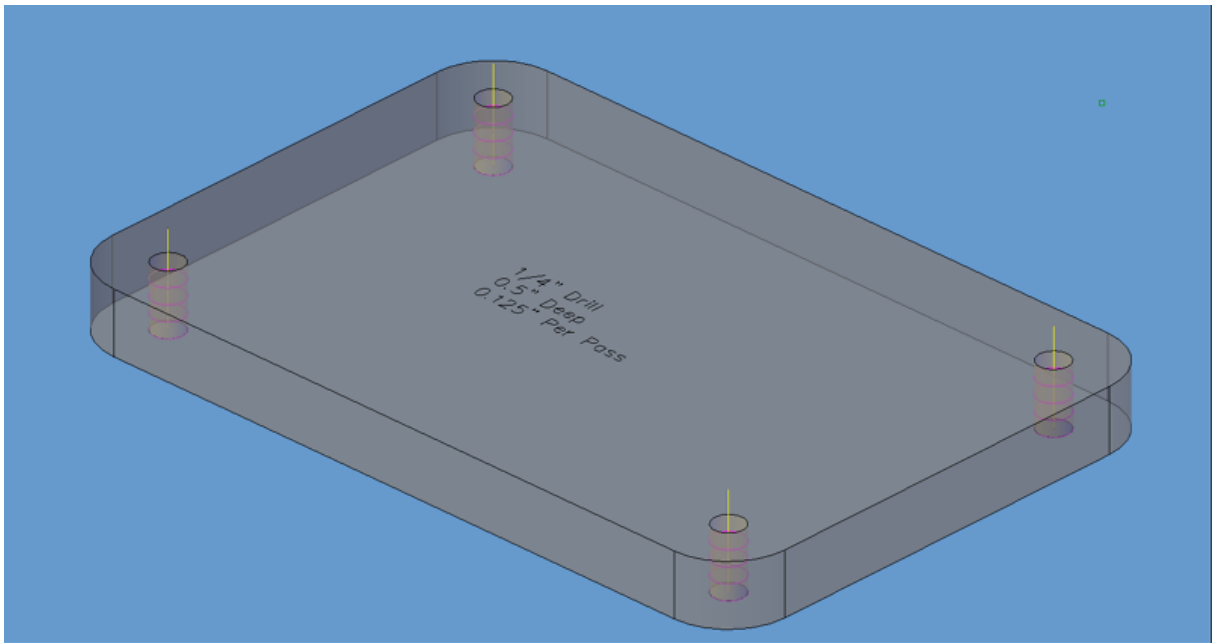
Drill Motions




The Drill Motions cycle is the standard drilling cycle in Router-CIM. With Drill Motions, you can drill in one pass, or in several passes with pecking motions.

By default the tool will rapid to the safety plane above the hole, then feed in to the depth of the cut and then rapid out of the hole back up to the safety plane. If you wish to peck drill the hole, you can set multiple depths per pass (peck increments) and the tool will rapid to the hole, feed down to the first depth, the rapid back to the safety plane, then rapid back down to .1 above the next material to be cut, then feed down to the next depth of cut, rapid up to the safety plane, etc. until it finishes the hole.

The drill motions cycle has very few parameters that can be changed as it is the most basic drilling cycle in Router-CIM.



Drill Motions cut cycle.

Cycle Information	Status Information
	Safety Plane <input type="text" value="0.25000"/>
	Depth Per Pass <input type="text" value="1.00000"/>
	Total Cut Depth <input type="text"/>
	Feedrate/Spindle Speed
	Feedrate <input type="text" value="350.00000"/>
	Spindle Speed <input type="text" value="18000.00000"/>
	Surface FPM <input type="text" value="NONE"/>
	Units Per Revolution <input type="text" value="NONE"/>
	<input type="button" value="Calculate"/>
	Before Codes <input type="text"/>
After Codes <input type="text"/>	
Oscillation Amount <input type="text" value="0.00000"/>	
Sort By Rank # <input type="text"/>	
	
<input type="button" value="Reset Cycle Settings to Default"/>	

Drill Motions Cut Cycle Parameters

The following parameters effect the toolpath creation:

Adding a Dwell/Pause at the end of a Drill Cycle

To add a dwell/pause at the bottom of a Drill Motions cycle, you will need to adjust the following setting in the Drill Motion cycle:

To make this change, go to the Router-CIM Control panel in AutoCAD and select the button for **'Mod Cycle'**.

Change Position 17, '**Task @ Cut End (name/N)**' in the Drill Motions cycle to DWELL/1 (or the time you want to pause, in seconds). You will get a G04P1 in the code after the tool is at cut depth.

Alternately on newer posts, you can use DWELLX.1 and get G04X.1.

To make this change, select Position 17, '**Task @ Cut End (name/N)**', type in the new value as stated above and hit '**Enter**'. You should see the value update in Position 17.

Edit Parameters

Parametric name: ROUTER_DRILL_MOTIONS

☐ Icon

Description	Value
[12]Lead-Out Name ("NAME"/N)	n
[13]Size of Leads.....(#)	leadscl
[14]Angle of Leads.....(#/N)	0.0
[15]Task @ Lead-In (name/N)	driplu
[16]Task @ Cut Start(name/N)	nothick
[17]Task @ Cut End (name/N)	n
[18]Task @ Lead-Out (name/N)	retract4
[19].....	
[20]Description.....("Text")	"Drill motions"

Edit Value of Selected Parameter

Task @ Cut End (name/N) ☒

OK Cancel Notes.. TaskInfo... Write...

Permanent Editing

Note: To know which option is right for you, please review your Post Processor's Application Notes.

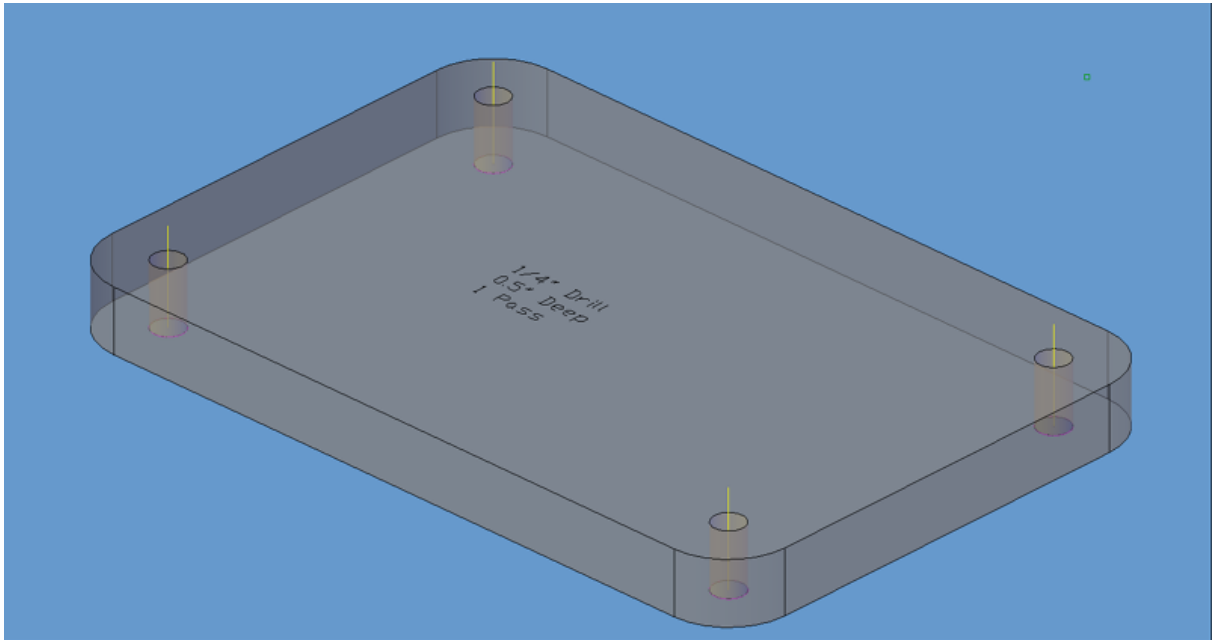
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Fast Drill




There is a Fast Drill cycle included with Router-CIM. Not all machines support this feature, but most Fanuc controlled machines will. The basic idea is to remove the dwell caused by switching from rapid to a programmed feedrate, and back to rapid again on each hole. This causes a small dwell on each change from G0 to G1. Instead the code will keep the machine in G1 mode and change the feedrate from the speed programmed for the drill to the fastest feedrate available for the machine on the moves where there would normally be a rapid traverse move. This causes the machine to move much faster from hole to hole, by avoiding the dwell.

In use the cycle appears the same as Drill Motions, but the resulting NC Code in the program is very different.



Fast Drill cut cycle.

Cycle Information		Status Information	
Index Speed	<input type="text" value=""/>	Safety Plane	<input type="text" value="*0.25000"/>
Fastdrill	<input type="text" value="Y"/>	Depth Per Pass	<input type="text" value="0.20000"/>
		Total Cut Depth	<input type="text" value="-0.75000"/>
		Feedrate/Spindle Speed	
		Feedrate	<input type="text" value="1000.00000"/>
		Spindle Speed	<input type="text" value="18000.00000"/>
		Surface FPM	<input type="text" value="NONE"/>
		Units Per Revolution	<input type="text" value="NONE"/>
		<input type="button" value="Calculate"/>	
		Before Codes	<input type="text" value=""/>
		After Codes	<input type="text" value=""/>
		Oscillation Amount	<input type="text" value="0.00000"/>
		Sort By Rank #	<input type="text" value=""/>
			
		<input type="button" value="Reset Cycle Settings to Default"/>	

Fast-Drill Cut Cycle Parameters

The following parameters effect the toolpath creation:

Index Speed

This is the fastest speed that you want the machine to achieve between the drilled holes. This feedrate will take place of the rapid traverse move between cuts and during the retract of the hole. If the machine can make a fast linear move between the cuts, usually this will reduce the overall cycle time of the drill moves.

Fastdrill

This engages the fast drill cycle. Entering Y will turn it on. Entering N will turn it off and the g-code will revert to the 'Drill Motions' cycle.

Adding a Dwell/Pause at the end of a Drill Cycle

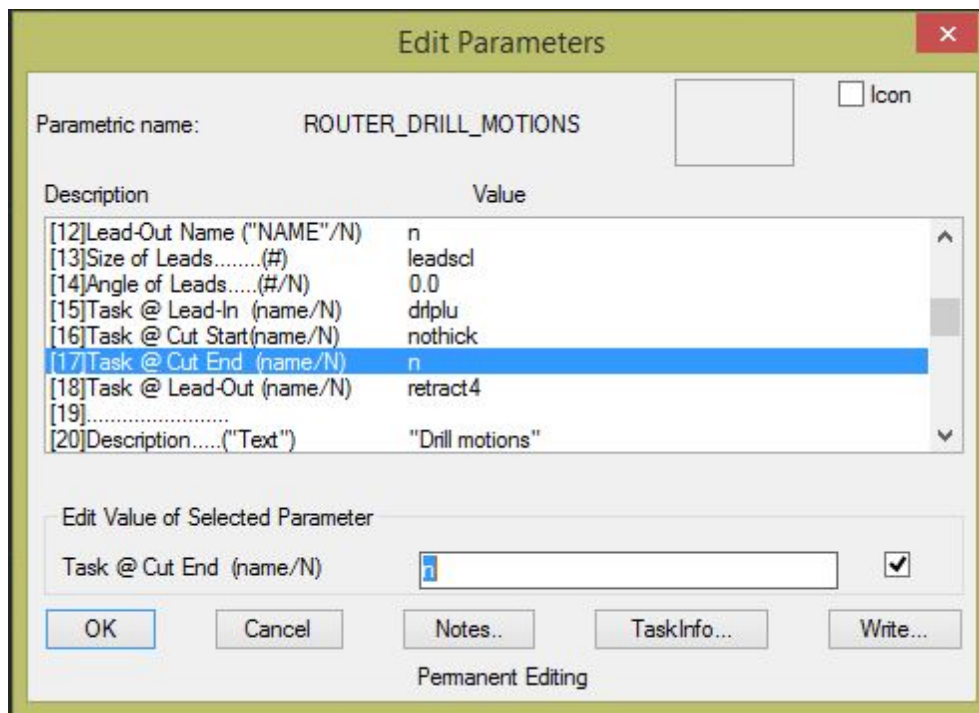
To add a dwell/pause at the bottom of a Drill Motions/Fast Drill cycle, you will need to adjust the following setting in the Drill Motions/Fast Drill cycle:

To make this change, go to the Router-CIM Control panel in AutoCAD and select the button for '**Mod Cycle**'.

Change Position 17, '**Task @ Cut End (name/N)**' in the Drill Motions cycle to DWELL/1 (or the time you want to pause, in seconds). You will get a G04P1 in the code after the tool is at cut depth.

Alternately on newer posts, you can use DWELLX.1 and get G04X.1.

To make this change, select Position 17, '**Task @ Cut End (name/N)**', type in the new value as stated above and hit '**Enter**'. You should see the value update in Position 17.



Note: To know which option is right for you, please review your Post Processor's Application Notes.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

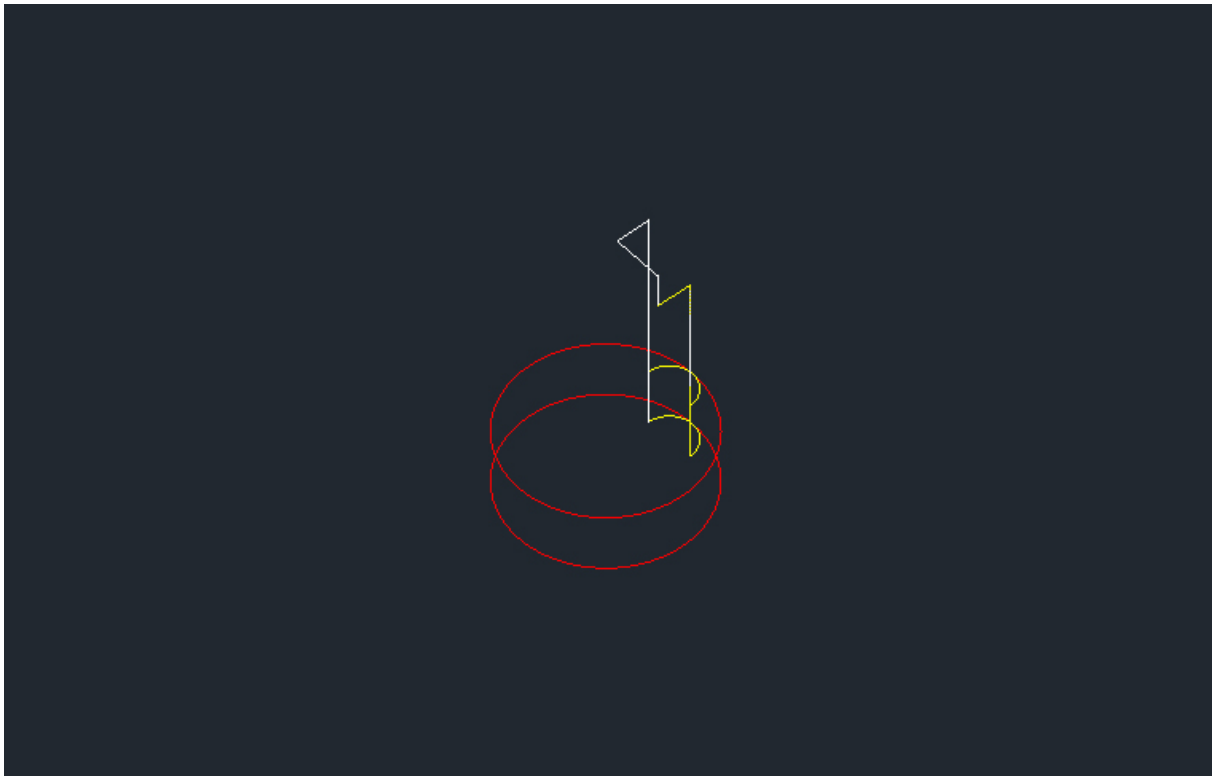
Drill-Interpolation



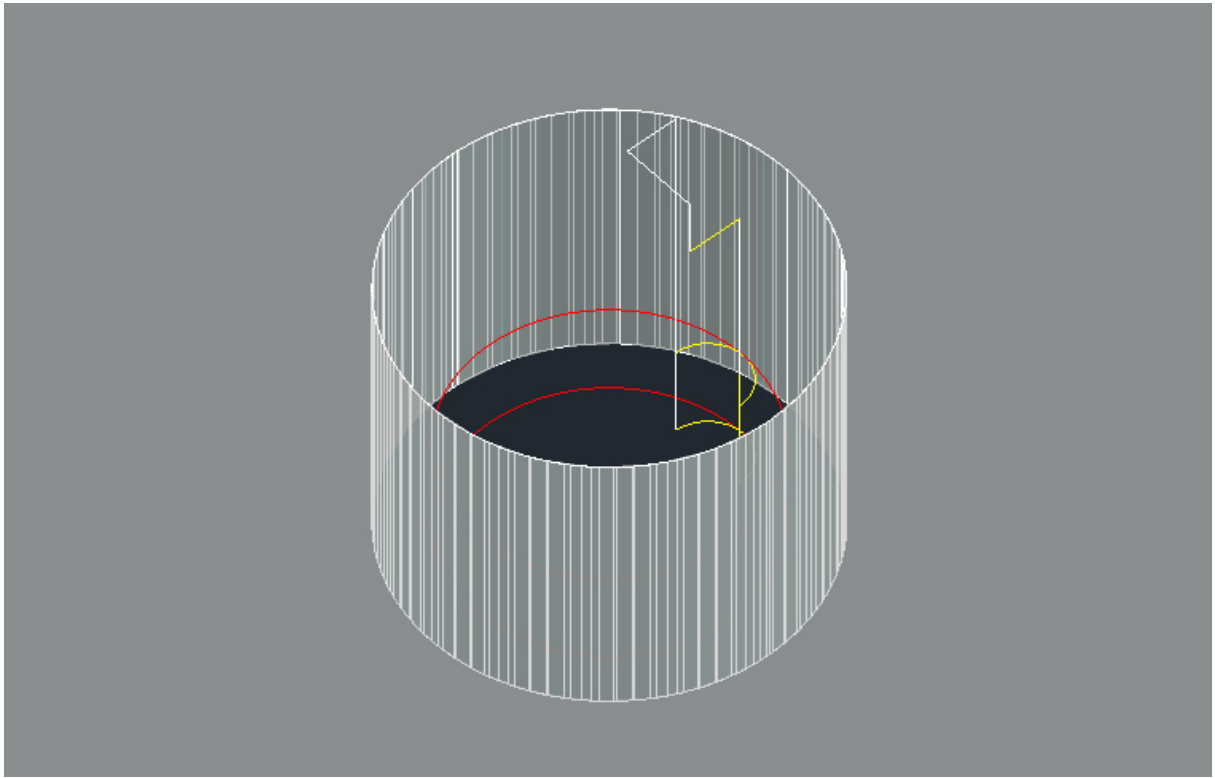
Drill Interpolation is a somewhat misleading term. This is actually a ROUTING cycle, but was created to allow for a way to cut a hole when there was no drill bit of the correct size available and there was a router-bit that was some amount smaller than the hole to be cut.

It is important to note that the router-bit should be smaller than the hole. It is also important to note that you should use a somewhat slower speed to interpolate the hole, or you may end up with an egg shaped hole.


Sometimes it is helpful to set the safety plane a little higher if you are using Cutter Compensation to allow the machine time to make the cutter comp move before it starts moving the cutter down into the hole to interpolate.



Drill-Interpolation Tool Path



Drill-Interpolation Cut Cycle

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▾	Safety Plane	*0.25000
Cut Direction	CCW ▾	Depth Per Pass	1.00000
Lead In	LI ▾	Total Cut Depth	
Lead Out	LO ▾	Feedrate/Spindle Speed	
Lead Size	0.01 ▾	Feedrate	350.00000
Lead Feed		Spindle Speed	18000.00000
Overlap Amount	AUTO ▾	Surface FPM	NONE
XY Stock Allowance		Units Per Revolution	NONE
Z Stock Allowance		<input type="button" value="Calculate"/>	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		<input type="button" value="Reset Cycle Settings to Default"/>	

Drill-Interpolation Cut Cycle Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (OFFSZ) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

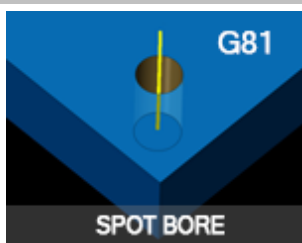
Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Spot-Bore



Spot Bore is a Canned Cycle. That really means that it is a text based cycle that has several parameters which are fed to the machine controller so that all the holes chosen are cut in exactly the same manner, like a macro. In fact on most machines, Canned Cycles are an option and you should check your machine to see if it is equipped with this cycle before trying to use it. Typically, Canned Cycles are used to lessen the amount of NC Code necessary in a program when drilling a large amount of holes. There is a code savings because the Z moves up and down in each hole are not necessary to call out in the code. Another reason to use Canned Cycles is if you are writing code by hand, as there is less code to generate. Finally, if there is a chance that you are going to edit the code later by hand, a Canned Cycle leaves you with only one line to edit to change all the drilling parameters.

Spot Bore is typically a normal one-pass drilling operation performed to the bottom of the hole.

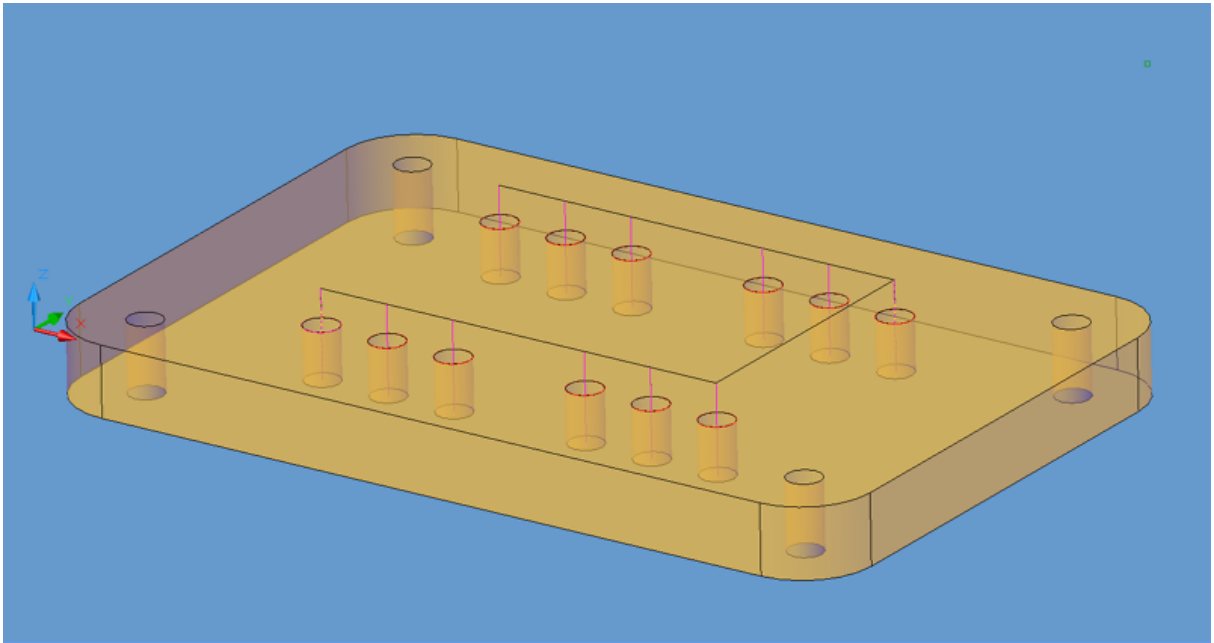
Canned Cycles use a series of commands on the first hole chosen and then perform those same commands on all other holes in the group. To use Canned Cycles, some explanation of them is necessary.

Note: This data is from the Fanuc control manual and will be specific to Fanuc controls or controls that can emulate the Fanuc code.

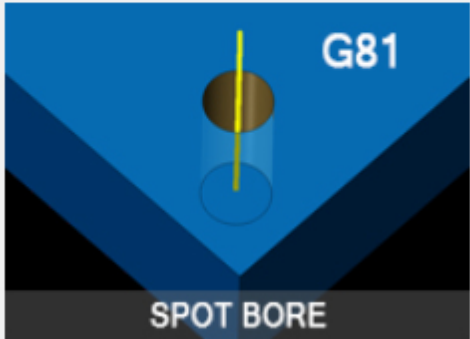
Format

G81 X_ Y_ Z_ R_ F_ K_ ;	
X_ Y_ : Hole position data Z_ : The distance from point R to the bottom of the hole R_ : The distance from the initial level to point R level F_ : Cutting feedrate K_ : Number of repeats	
G81 (G98)	G81 (G99)

G81 Canned Cycle



Holes drilled with Canned Cycles

Cycle Information		Status Information	
Type	81	Safety Plane	*0.25000
Mode	98	Depth Per Pass	1.00000
Retract		Total Cut Depth	
		Feedrate/Spindle Speed	
		Feedrate	350.00000
		Spindle Speed	18000.00000
		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Canned Cycle Parameters

The following parameters effect the toolpath creation:

Type

The type field for Canned Cycles indicated the Cycle Type or method of the canned cycle. In this case G81 is the desired Canned Cycle, so 81 is the answer in Type.

You can substitute any valid Canned Cycle type in this field to create code for another cycle type.

Mode

Mode refers to the Retract mode of the canned cycle. In the pictures above, there is either G98 Mode, where the tool retracts to the Initial Point, or G99 Mode, where the tool retracts to the location specified by the R value (the next parameter).

R

This is the absolute point where you want the tool to retract to during pecks, between holes, and when the cycle is finished.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Peck-Bore



Peck Bore is a Canned Cycle. That really means that it is a text based cycle that has several parameters which are fed to the machine controller so that all the holes chosen are cut in exactly the same manner, like a macro. In fact on most machines, Canned Cycles are an option and you should check your machine to see if it is equipped with this cycle before trying to use it. Typically, Canned Cycles are used to lessen the amount of NC Code necessary in a program when drilling a large amount of holes. There is a code savings because the Z moves up and down in each hole are not necessary. Another reason to use Canned Cycles is if you are writing code by hand, as there is less code to generate. Finally, if there is a chance that you are going to edit the code later by hand, a Canned Cycle leaves you with only one line to edit to change all the drilling parameters.

Peck Bore is typically a multiple depth per pass drilling operation.

Canned Cycles use a series of commands on the first hole chosen and then perform those same commands on all other holes in the group. To use Canned Cycles, some explanation of them is necessary.

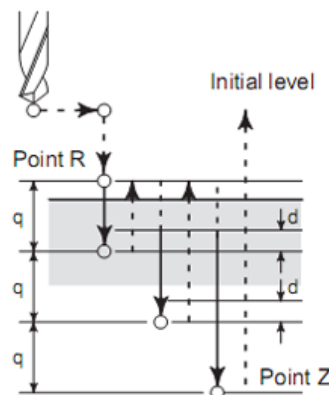
Note: This data is from the Fanuc control manual and will be specific to Fanuc controls or controls that can emulate the Fanuc code.

Format

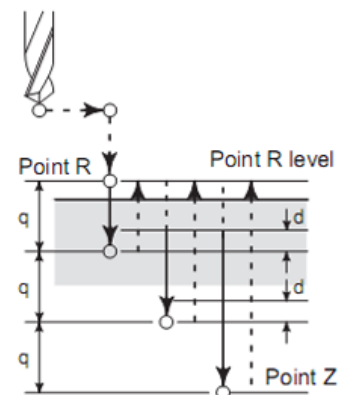
G83 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_ Y_ : Hole position data
 Z_ : The distance from point R to the bottom of the hole
 R_ : The distance from the initial level to point R level
 Q_ : Depth of cut for each cutting feed
 F_ : Cutting feedrate
 K_ : Number of repeats

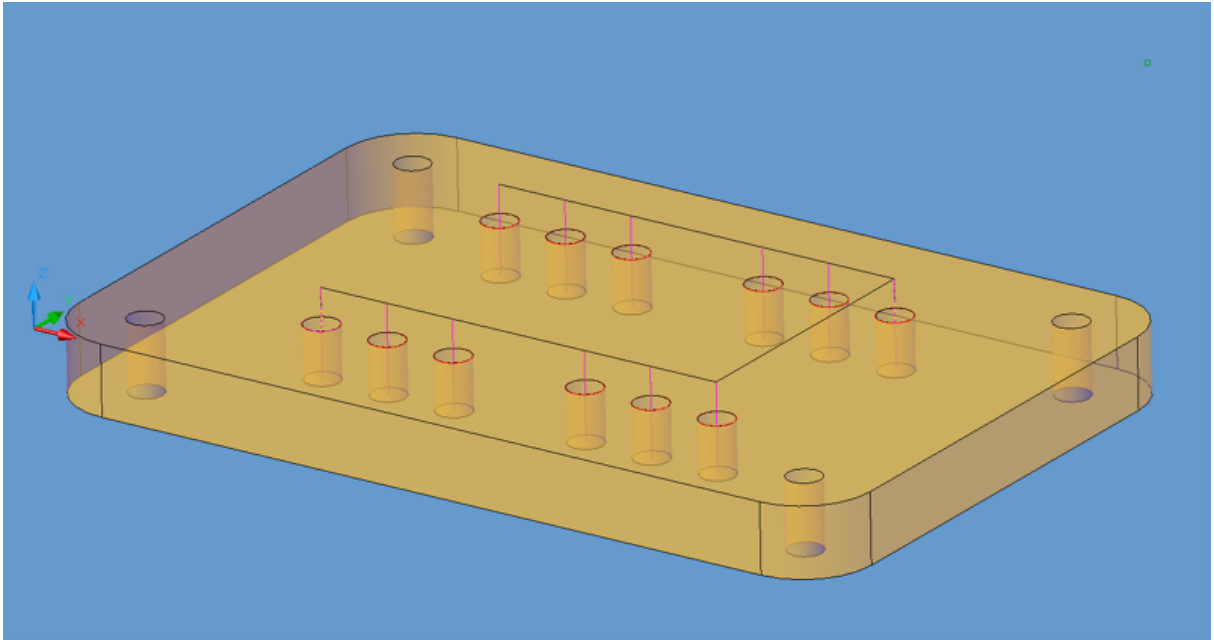
G83 (G98)




G83 (G99)



Peck Bore Cycle.



Holes drilled with Canned Cycles.

Cycle Information		Status Information	
Type	83	Safety Plane	*0.25000
Mode	98	Depth Per Pass	1.00000
Retract		Total Cut Depth	
		Feedrate/Spindle Speed	
		Feedrate	350.00000
		Spindle Speed	18000.00000
		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Peck Bore Parameters

The following parameters effect the toolpath creation:

Type

The type field for Canned Cycles indicated the Cycle Type or method of the canned cycle. In this case G81 is the desired Canned Cycle, so 81 is the answer in Type.

You can substitute any valid Canned Cycle type in this field to create code for another cycle type.

Mode

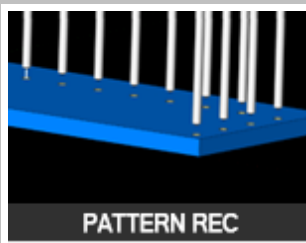
Mode refers to the Retract mode of the canned cycle. In the pictures above, there is either G98 Mode, where the tool retracts to the Initial Point, or G99 Mode, where the tool retracts to the location specified by the R value (the next parameter).

Retract

This is the absolute point where you want the tool to retract to during pecks, between holes, and when the cycle is finished.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

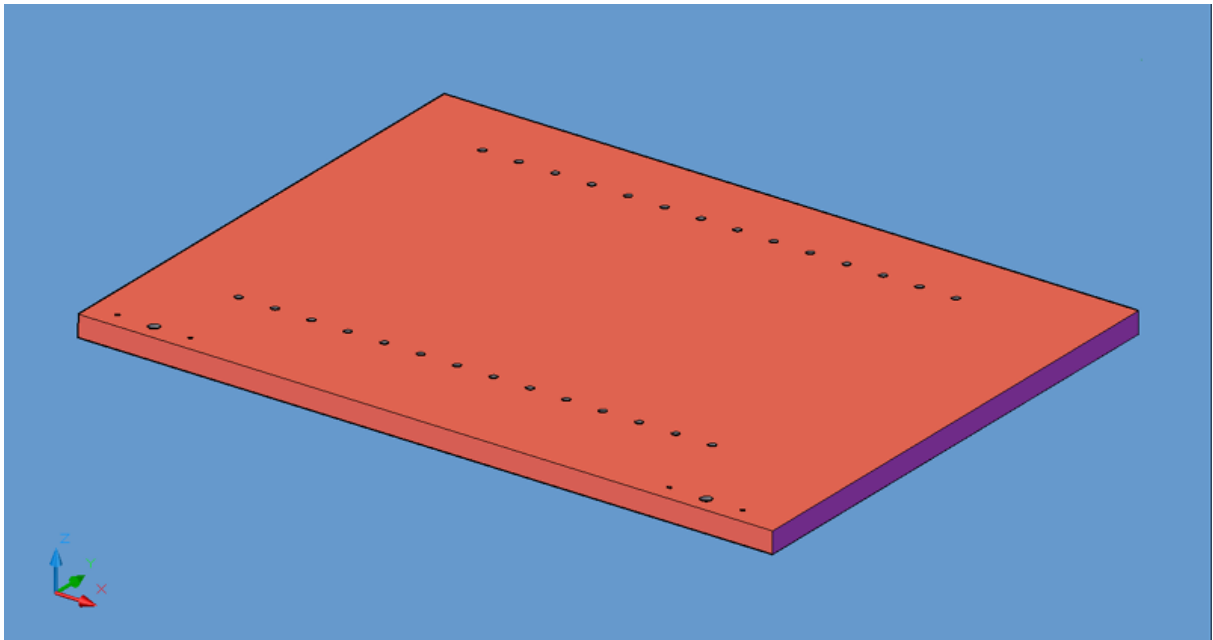
Pattern Recognition



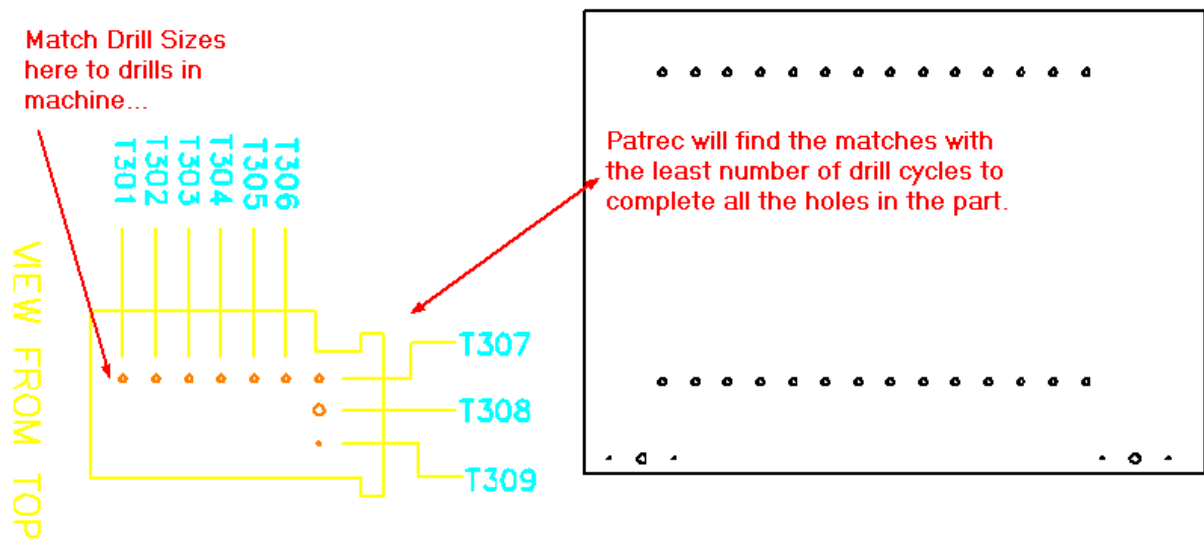
The Pattern Recognition cycle examines the drill holes on your part and the configuration of your drilling head(s) to determine the most efficient means of boring the selected holes. In order to use this function properly, the configuration of your drill heads must be set before you attempt to use this cycle. (See Note below.) Once configured, pick Pattern Recognition from the Cycle menu, choose Cut, and then select the holes to be bored. Pattern Recognition will load into memory, determine the most efficient route, and then proceed to bore the holes. The lead spindle in the group will have the tool path displayed on the screen; the others in the group will change colors to show they have been processed. The gang drill drawing will be provided to you by CIM-Tech following the specifications of your machine tool. Drill holes processed with Pattern Recognition need not be Geoshaped, and should be given thickness equal to their depth (negative Z value).

Note: The size of the holes on your gang drill drawing in your machine-specific default drawing **MUST BE THE SAME SIZE** as the holes in your current drawing for Pattern Recognition to work. You do not need to add any tool numbers in the Control Panel, as Pattern Recognition will find the tools it needs and add them automatically.

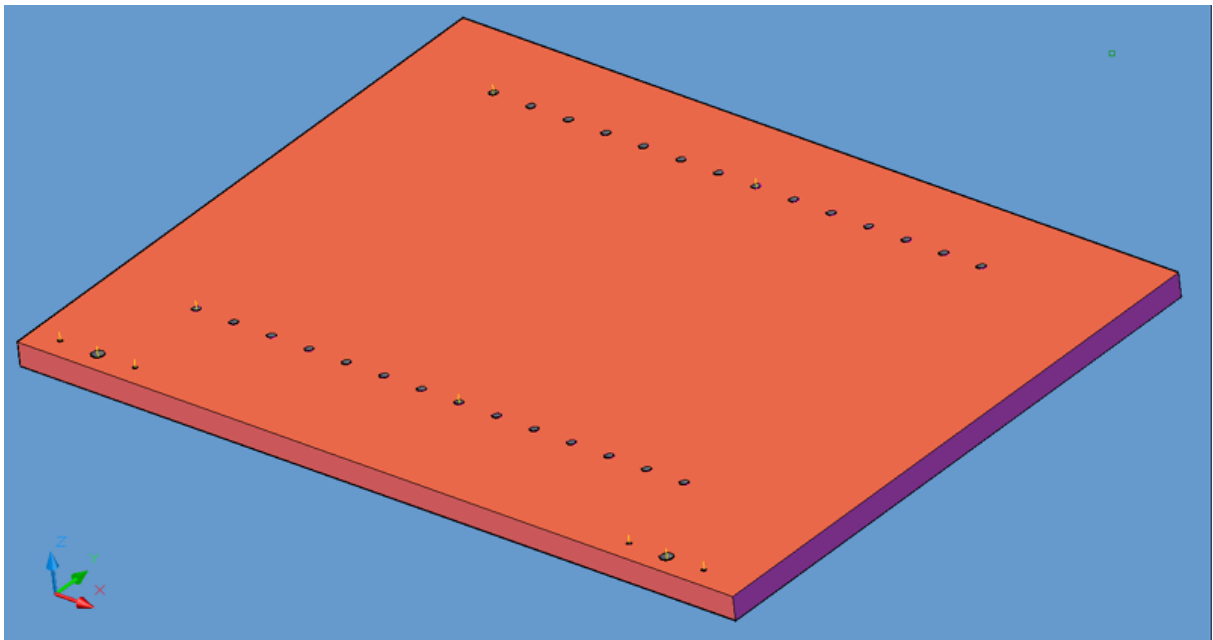
Note: Pattern Recognition is only available on certain machine types that allow for an optimized boring block. Please review your Post Processor's Application Notes to see if this cycle is available with your machine.



Typical Pattern Recognition Part.

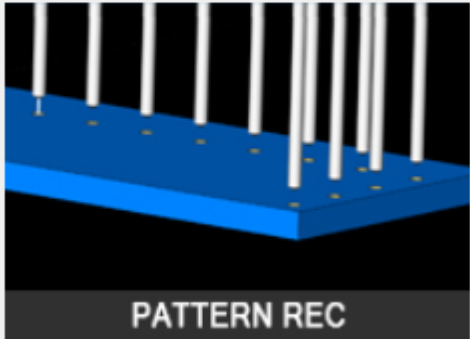


Match the drill block drawing to the drills in the machine.



Result of Pattern Recognition.

In the example above, the two rows of shelf holes are 32mm apart (1.2598") and there are two rows of 14 each. There are also two sets of hardware holes with sizes that do not match the shelf holes. With the drill block shown above, there are 7 drills in a row that match the shelf holes, so the drill block only drops twice on each row, drilling 7 holes each time. It then drills each of the hardware holes one at a time. In this instance, there are 10 drill cycles for a total of 34 holes.

Cycle Information	Status Information
	Safety Plane <input type="text" value="*0.25000"/>
	Depth Per Pass <input type="text" value="1.00000"/>
	Total Cut Depth <input type="text"/>
	Feedrate/Spindle Speed
	Feedrate <input type="text" value="350.00000"/>
	Spindle Speed <input type="text" value="18000.00000"/>
	Surface FPM <input type="text" value="NONE"/>
	Units Per Revolution <input type="text" value="NONE"/>
	<input type="button" value="Calculate"/>
	Before Codes <input type="text"/>
After Codes <input type="text"/>	
Oscillation Amount <input type="text" value="0.00000"/>	
Sort By Rank # <input type="text"/>	
	
<input type="button" value="Reset Cycle Settings to Default"/>	

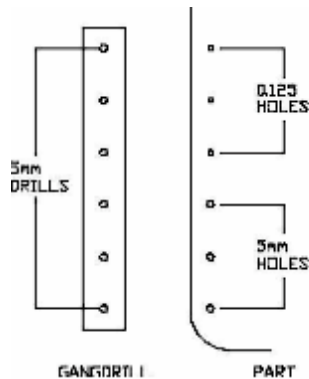
Pattern Recognition Parameters

Pattern Recognition - Tolerance Variable (*pat_fuzz*)

For Pattern Recognition to work properly, the diameter of the holes in your current drawing must be the same size as the holes on your gang drill drawing in your machine-specific default drawing (which is loaded into your current drawing upon loading Router-CIM). There is a tolerance built into the system, for the diameter of the holes, that is set to 0.05. This variable is *pat_fuzz* in the NCVARS. Under normal circumstances this need not be changed. But, if you have tools of a similar size (i.e. within .05 in radius or diameter) then Pattern Recognition may choose the wrong tools to complete its boring operation. Generally, if you are using all inch or all metric this will not be a problem.

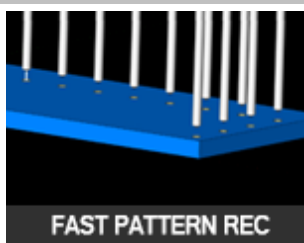
Example: There are six drills on the gang drill with 5mm diameter drills in them. The part has both 5mm and 0.125 diameter holes in it. The difference between these two diameters is within the 0.05 tolerance,

so Pattern Recognition will see them all as one size. Therefore, all six drills will drop and drill all six holes and all hole diameters will be 5mm, which is incorrect.



****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Fast Pattern Recognition

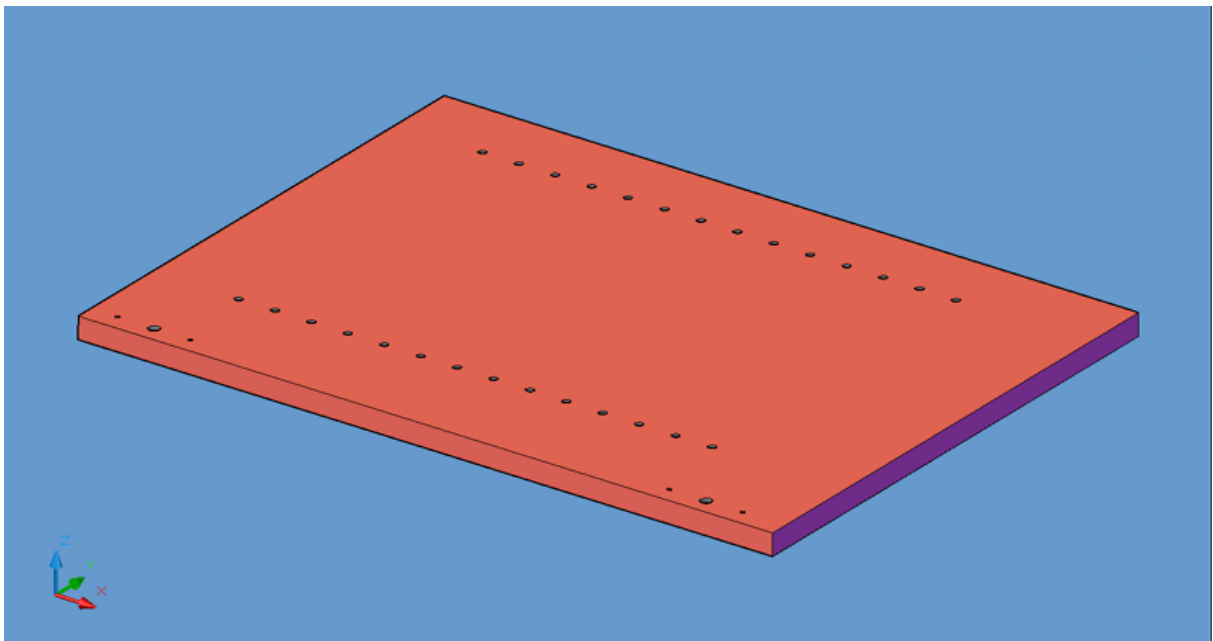


Fast Pattern Recognition is the same as regular Pattern Recognition, except that it substitutes fast feed moves for the rapid index moves between drills, which actually speeds up the overall drilling operations.

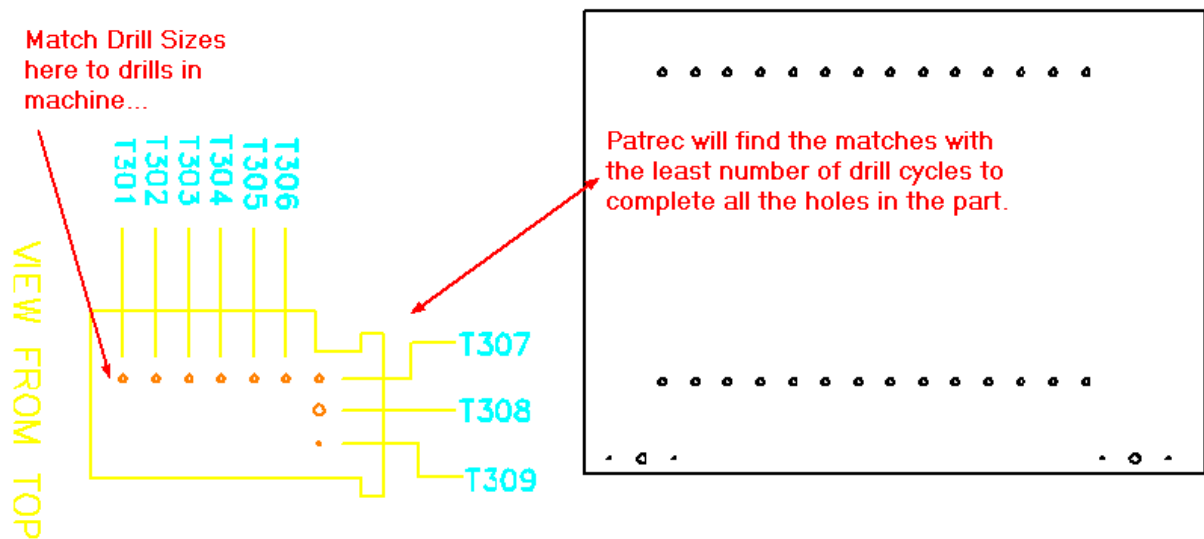
The Pattern Recognition cycle examines the drill holes on your part and the configuration of your drilling head(s) to determine the most efficient means of boring the selected holes. In order to use this function properly, the configuration of your drill heads must be set before you attempt to use this cycle. (See Note below.) Once configured, pick Pattern Recognition from the Cycle menu, choose Cut, and then select the holes to be bored. Pattern Recognition will load into memory, determine the most efficient route, and then proceed to bore the holes. The lead spindle in the group will have the tool path displayed on the screen; the others in the group will change colors to show they have been processed. The gang drill drawing will be provided to you by CIM-Tech following the specifications of your machine tool. Drill holes processed with Pattern Recognition need not be Geoshaped, and should be given thickness equal to their depth (negative Z value).

Note: The size of the holes on your gang drill drawing in your machine-specific default drawing **MUST BE THE SAME SIZE** as the holes in your current drawing for Pattern Recognition to work. You do not need to add any tool numbers in the Control Panel, as Pattern Recognition will find the tools it needs and add them automatically.

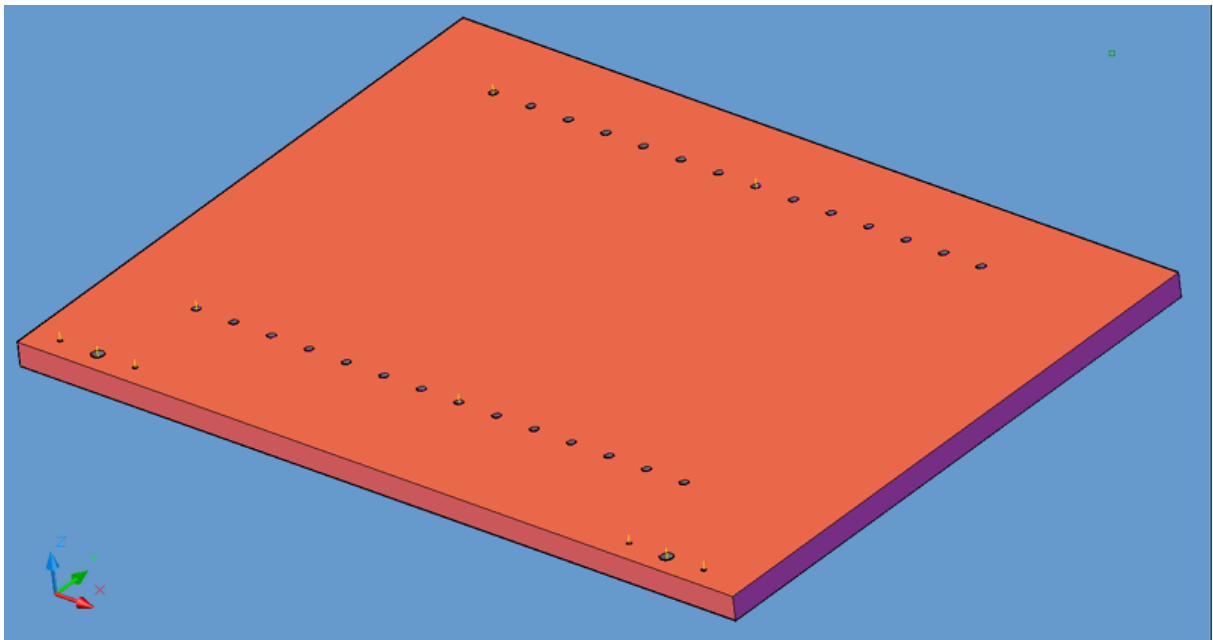
Note: Pattern Recognition is only available on certain machine types that allow for an optimized boring block. Please review your Post Processor's Application Notes to see if this cycle is available with your machine.



Typical Pattern Recognition Part.

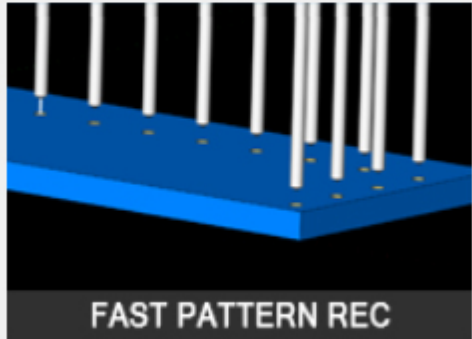


Match the drill block drawing to the drills in the machine.



Result of Pattern Recognition.

In the example above, the two rows of shelf holes are 32mm apart (1.2598") and there are two rows of 14 each. There are also two sets of hardware holes with sizes that do not match the shelf holes. With the drill block shown above, there are 7 drills in a row that match the shelf holes, so the drill block only drops twice on each row, drilling 7 holes each time. It then drills each of the hardware holes one at a time. In this instance, there are 10 drill cycles for a total of 34 holes.

Cycle Information		Status Information	
Index Speed	<input type="text"/>	Safety Plane	<input type="text" value="*0.25000"/>
Fastdrill	<input type="text" value="Y"/>	Depth Per Pass	<input type="text" value="0.50000"/>
		Total Cut Depth	<input type="text" value="-0.75000"/>
		Feedrate/Spindle Speed	
		Feedrate	<input type="text" value="1000.00000"/>
		Spindle Speed	<input type="text" value="18000.00000"/>
		Surface FPM	<input type="text" value="NONE"/>
		Units Per Revolution	<input type="text" value="NONE"/>
		<input type="button" value="Calculate"/>	
		Before Codes	<input type="text"/>
		After Codes	<input type="text"/>
		Oscillation Amount	<input type="text" value="0.00000"/>
		Sort By Rank #	<input type="text"/>
			
		<input type="button" value="Reset Cycle Settings to Default"/>	

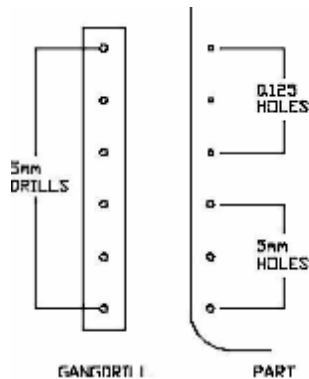
Fast Pattern Recognition Parameters

Pattern Recognition - Tolerance Variable (*pat_fuzz*)

For Pattern Recognition to work properly, the diameter of the holes in your current drawing must be the same size as the holes on your gang drill drawing in your machine-specific default drawing (which is loaded into your current drawing upon loading Router-CIM). There is a tolerance built into the system, for the diameter of the holes, that is set to 0.05. This variable is *pat_fuzz* in the NCVARS. Under normal circumstances this need not be changed. But, if you have tools of a similar size (i.e. within .05 in radius or diameter) then Pattern Recognition may choose the wrong tools to complete its boring operation. Generally, if you are using all inch or all metric this will not be a problem.

Example: There are six drills on the gang drill with 5mm diameter drills in them. The part has both 5mm and 0.125 diameter holes in it. The difference between these two diameters is within the 0.05 tolerance,

so Pattern Recognition will see them all as one size. Therefore, all six drills will drop and drill all six holes and all hole diameters will be 5mm, which is incorrect.



The following parameters effect the toolpath creation:

Index Speed

This is the fastest speed that you want the machine to achieve between the drilled holes. This feedrate will take place of the rapid traverse move between cuts and during the retract of the hole. If the machine can make a fast linear move between the cuts, usually this will reduce the overall cycle time of the drill moves.

Fastdrill

This engages the fast drill cycle. Entering Y will turn it on. Entering N will turn it off and the g-code will revert to the 'Drill Motions' cycle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Deep-Bore

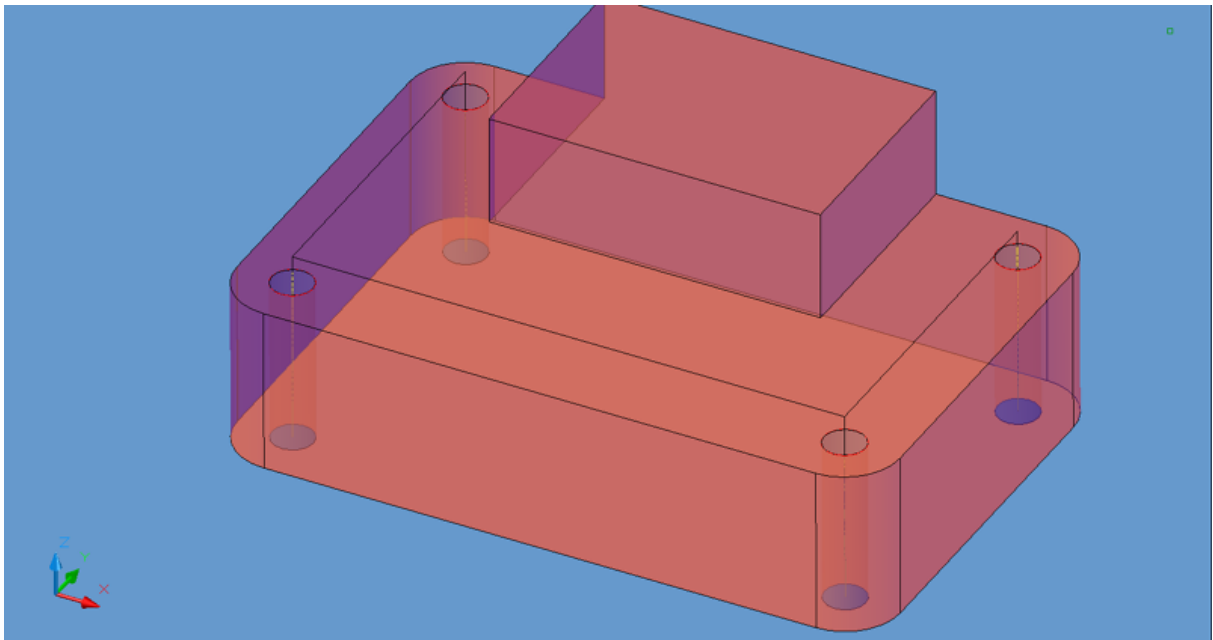


Deep Bore is a multiple pass drilling cycle that offers some flexibility with regards to peck increments, chip clearing, and changing feedrates during the cut cycle.

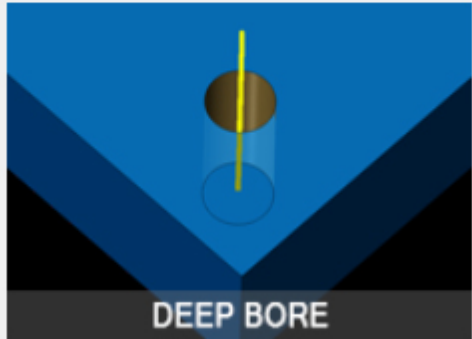
Typically this cycle will start at the Safety Plane, drill down to the first peck depth, retract enough to break the chip and then bore to the next peck depth. There is a parameter setting for the Chip Clear Increment and Chip Break Height where the tool can retract to the clearance plane between pecks to remove chips from the hole. There are also parameters available to change the feedrate at a certain point during the cut and allowances to change the peck increment at a specific depth, as well as new chip break and chip clear increments.

This cut cycle is meant for drilling in very thick material or when the tool needs to be retracted several times to keep the chips from clogging the tool.

Holes drilled with this cycle should be grouped together like canned cycles so that each cut task can run on each hole.



Deep Bore Cycle

Cycle Information	Status Information
Chip Break Height <input type="text"/>	Safety Plane <input type="text" value="0.25000"/>
Chip Clear Increment <input type="text"/>	Depth Per Pass <input type="text" value="1.00000"/>
Feed Change Depth <input type="text"/>	Total Cut Depth <input type="text"/>
New Feedrate <input type="text"/>	Feedrate/Spindle Speed
Peck Change Depth <input type="text"/>	Feedrate <input type="text" value="350.00000"/>
New Peck Increment <input type="text"/>	Spindle Speed <input type="text" value="18000.00000"/>
New Chip Break Height <input type="text"/>	Surface FPM <input type="text" value="NONE"/>
New Clear Increment <input type="text"/>	Units Per Revolution <input type="text" value="NONE"/>
	<input type="button" value="Calculate"/>
	Before Codes <input type="text"/>
	After Codes <input type="text"/>
	Oscillation Amount <input type="text" value="0.00000"/>
	Sort By Rank # <input type="text"/>
	
	<input type="button" value="Reset Cycle Settings to Default"/>

Deep Bore Parameters

The following parameters effect the toolpath creation:

Chip Break Height

How far to back up on a chip break move. A chip break move is usually a short motion to fracture the chip, and does not clear the hole.

Chip Clear Increment

Usually a multiple of the peck increment. When this secondary increment is passed, the drill is retracted to the clear plane.

Chip Break and Chip Clear work together making short strokes until the drill is loaded, then clearing the drill and starting over.

Feed Change

Entered as a depth in Z (typically negative). Change to a new feedrate when this depth is reached or passed.

New Feedrate

New feedrate to change to when the Feed Change depth is reached or passed.

Peck Change

Depth to modify the Peck Increment, and chip handling values. Change to new pecking and chip handling values when this depth is reached or passed.

New Peck Increment

The value to use as the Peck Increment once the Peck Change depth is reached or passed.

New Chip Break

The value to use as the Chip Break Height once the Peck Change depth is reached or passed.

New Clear Increment

The new Chip Clear Increment to use once the Peck Change depth is reached or passed.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Helical-Arcs-Center

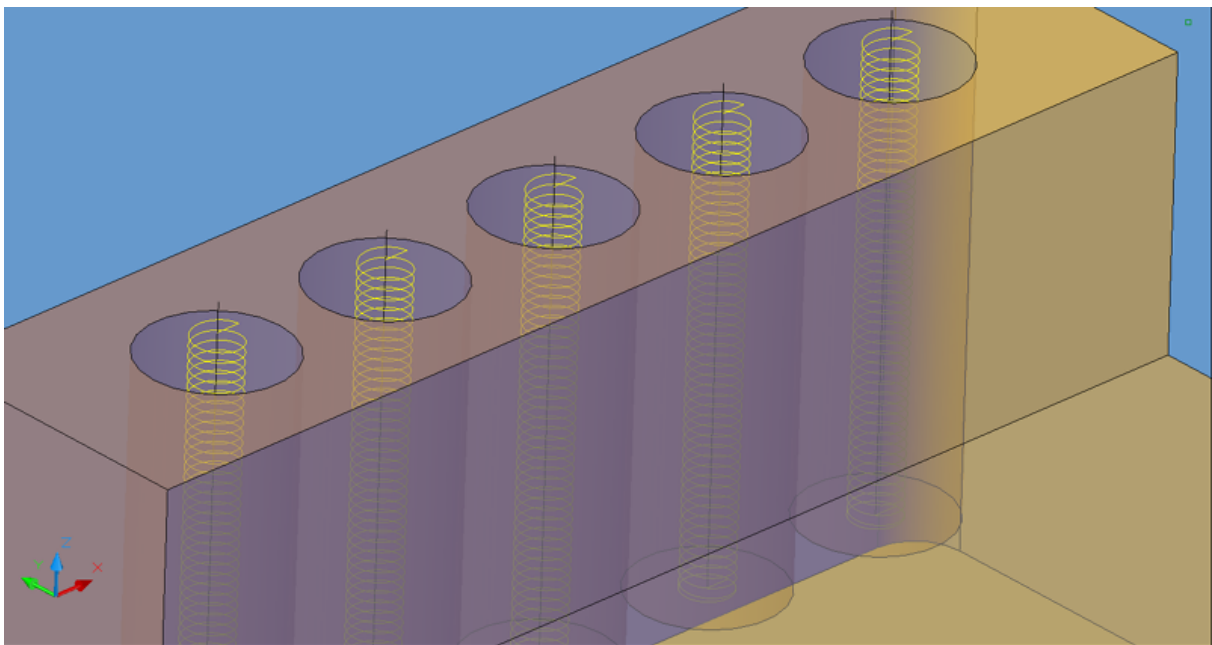


Helical Arcs can be used to make deep holes with a cutter as well as making threads with a thread mill. The benefit of making a regular hole cut with this cycle is that the tool chip load is constant and the tool does not switch back and forth between being loaded and unloaded, which minimizes tool deflection. There are parameters that allow you to have a bottom clean up pass so that there is no small wedge left at the bottom of the hole.

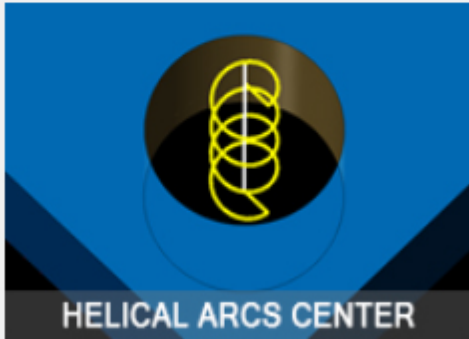
Thread milling can be accomplished with this cycle as well, by using a thread mill and entering in the parameters for the thread pitch. There are parameters that let you start at either the bottom or top of the hole.

Helical Arcs Center will start the tool in the center of the hole and move towards the edge in the lead-in move.

This cycle will produce X, Y and Z moves on an arc at the machine. The objects used to create this tool path are Geoshaped circles.



Helical Arcs Center tool path.

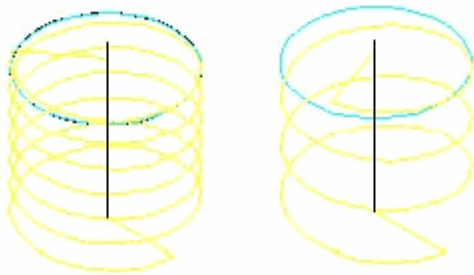
Cycle Information		Status Information	
Helix Pitch	!td*	Safety Plane	*0.25000
Helix Direction	CCW	Depth Per Pass	1.00000
External (Cut Side)		Total Cut Depth	
Bottom Cleanup		Feedrate/Spindle Speed	
Start At Top	Y	Feedrate	350.00000
Arc Radius In		Spindle Speed	18000.00000
Full Radius In		Surface FPM	NONE
Arc In Ramp	NONE	Units Per Revolution	NONE
Arc Radius Out		<input type="button" value="Calculate"/>	
Full Radius Out		Before Codes	
Arc Out Ramp	NONE	After Codes	
Cylinder Taper Angle		Oscillation Amount	0.00000
Cylinder Taper In		Sort By Rank #	
Start At Edge			
		<input type="button" value="Reset Cycle Settings to Default"/>	

Helical Arcs Center Parameters

The following parameters effect the toolpath creation:

Helix Pitch

The desired pitch, in units per one revolution. If you picture the spirals of the helix like threads on a screw, the Helix Pitch is the distance from one thread to the next.



Helix Direction

The rotation direction of the pitch, clockwise or counterclockwise. Inputs are either CW or CCW. If you are threading, this is the setting to make right or left hand threads.

External

Do an external or internal cut. The possible inputs for this are either Y or N (blank). Y will give you a tool path on the outside of the geoshaped geometry.

Bottom

This setting is for a Bottom Clean Up Pass in the cut. Possible inputs are Y or N. If Y is chosen, an extra cleanup pass is made at the bottom of the hole to remove excess material. This pass is only made at the Total Depth of Cut.

Start at Top

Y is the default. If No is chosen, the tool will feed to the bottom of a pre-cut hole and start its helical motion upward.

Arc Radius In

N will leave the lead-in arc radius at its default value. For a different lead-in radius, acceptable inputs for this field would be radius values in the form of a decimal (i.e. .25, .50, .60).

Full Radius In

Choose Y or N.

If Y is chosen, the cycle will have a 180° arc lead-in move based on the full radius of the tool.

Arc In Ramp

Choose Y or N.

If Y is selected, the cycle will have a helical lead-in.

Arc Radius Out

N will leave the lead-out arc radius at its default value. For a different lead-out radius, acceptable inputs for this field would be radius values in the form of a decimal (i.e. .25, .50, .60).

Full Radius Out

Choose Y or N.

If Y is chosen, the cycle will have a 180° arc lead-out move based on the full radius of the tool.

Arc Out Ramp

Choose Y or N.

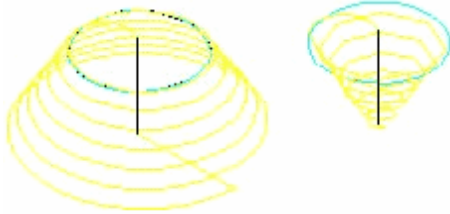
If Y is selected, the cycle will have a helical lead-out.

Cylinder Taper

Degrees of angle of cylinder taper. Possible inputs are positive whole number angles in degrees in 1 degree increments (i.e. 10, 20, 45 etc.).

Cylinder Taper IN

Choose Y or N. N will taper the cylinder outwards or bigger towards the bottom of the hole; Y will taper the cylinder in towards the bottom of the hole.

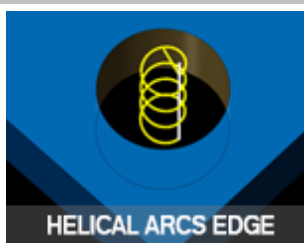


Start At Edge

Choose T or N. N will start in the center of the circle; T will start at the edge of the hole, 'Helical Arcs Edge' cutting cycle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Helical-Arcs-Edge

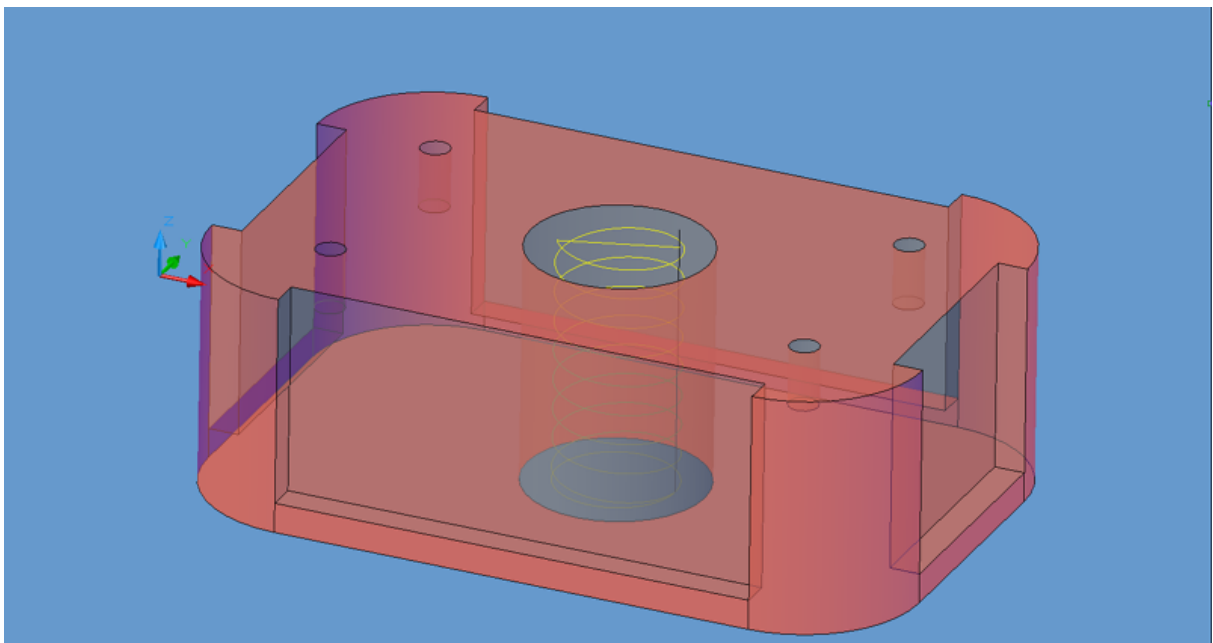


Helical Arcs can be used to make deep holes with a cutter as well as making threads with a thread mill. The benefit of making a regular hole cut with this cycle is that the tool chip load is constant and the tool does not switch back and forth between being loaded and unloaded, which minimizes tool deflection. There are parameters that allow you to have a bottom clean up pass so that there is no small wedge left at the bottom of the hole.


Thread milling can be accomplished with this cycle as well, by using a thread mill and entering in the parameters for the thread pitch. There are parameters that let you start at either the bottom or top of the hole.

Helical Arcs Edge will start the tool at the edge of the hole where the start point is chosen.

This cycle will produce X, Y and Z moves on an arc at the machine. The objects used to create this tool path are Geoshaped circles.



Helical Arcs Edge tool path.

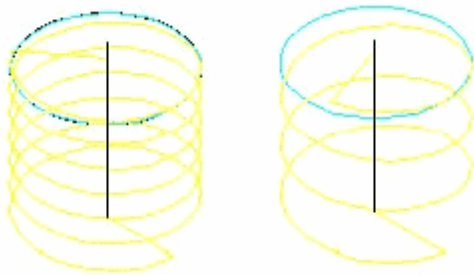
Cycle Information		Status Information	
Helix Pitch	!td*	Safety Plane	*0.25000
Helix Direction	CCW	Depth Per Pass	1.00000
External (Cut Side)		Total Cut Depth	
Bottom Cleanup		Feedrate/Spindle Speed	
Start At Top	Y	Feedrate	350.00000
Arc Radius In		Spindle Speed	18000.00000
Full Radius In		Surface FPM	NONE
Arc In Ramp	NONE	Units Per Revolution	NONE
Arc Radius Out		Calculate	
Full Radius Out		Before Codes	
Arc Out Ramp	NONE	After Codes	
Cylinder Taper Angle		Oscillation Amount	0.00000
Cylinder Taper In		Sort By Rank #	
Start At Edge	T		
		Reset Cycle Settings to Default	

Helical Arcs Edge Parameters

The following parameters effect the toolpath creation:

Helix Pitch

The desired pitch, in units per one revolution. If you picture the spirals of the helix like threads on a screw, the Helix Pitch is the distance from one thread to the next.



Helix Direction

The rotation direction of the pitch, clockwise or counterclockwise. Inputs are either CW or CCW. If you are threading, this is the setting to make right or left hand threads.

External

Do an external or internal cut. The possible inputs for this are either Y or N (blank). Y will give you a tool path on the outside of the geoshaped geometry.

Bottom

This setting is for a Bottom Clean Up Pass in the cut. Possible inputs are Y or N. If Y is chosen, an extra cleanup pass is made at the bottom of the hole to remove excess material. This pass is only made at the Total Depth of Cut.

Start at Top

Y is the default. If No is chosen, the tool will feed to the bottom of a pre-cut hole and start its helical motion upward.

Arc Radius In

N will leave the lead-in arc radius at its default value. For a different lead-in radius, acceptable inputs for this field would be radius values in the form of a decimal (i.e. .25, .50, .60).

Full Radius In

Choose Y or N.

If Y is chosen, the cycle will have a 180° arc lead-in move based on the full radius of the tool.

Arc In Ramp

Choose Y or N.

If Y is selected, the cycle will have a helical lead-in.

Arc Radius Out

N will leave the lead-out arc radius at its default value. For a different lead-out radius, acceptable inputs for this field would be radius values in the form of a decimal (i.e. .25, .50, .60).

Full Radius Out

Choose Y or N.

If Y is chosen, the cycle will have a 180° arc lead-out move based on the full radius of the tool.

Arc Out Ramp

Choose Y or N.

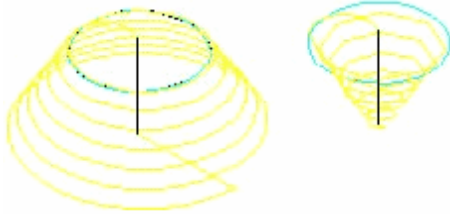
If Y is selected, the cycle will have a helical lead-out.

Cylinder Taper

Degrees of angle of cylinder taper. Possible inputs are positive whole number angles in degrees in 1 degree increments (i.e. 10, 20, 45 etc.).

Cylinder Taper IN

Choose Y or N. N will taper the cylinder outwards or bigger towards the bottom of the hole; Y will taper the cylinder in towards the bottom of the hole.

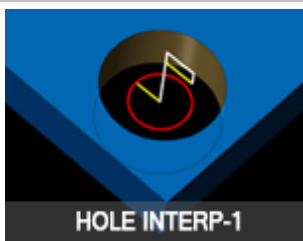


Start At Edge

Choose T or N. T will start at the edge of the hole; N will start in the center of the circle, 'Helical Arcs Center' cutting cycle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

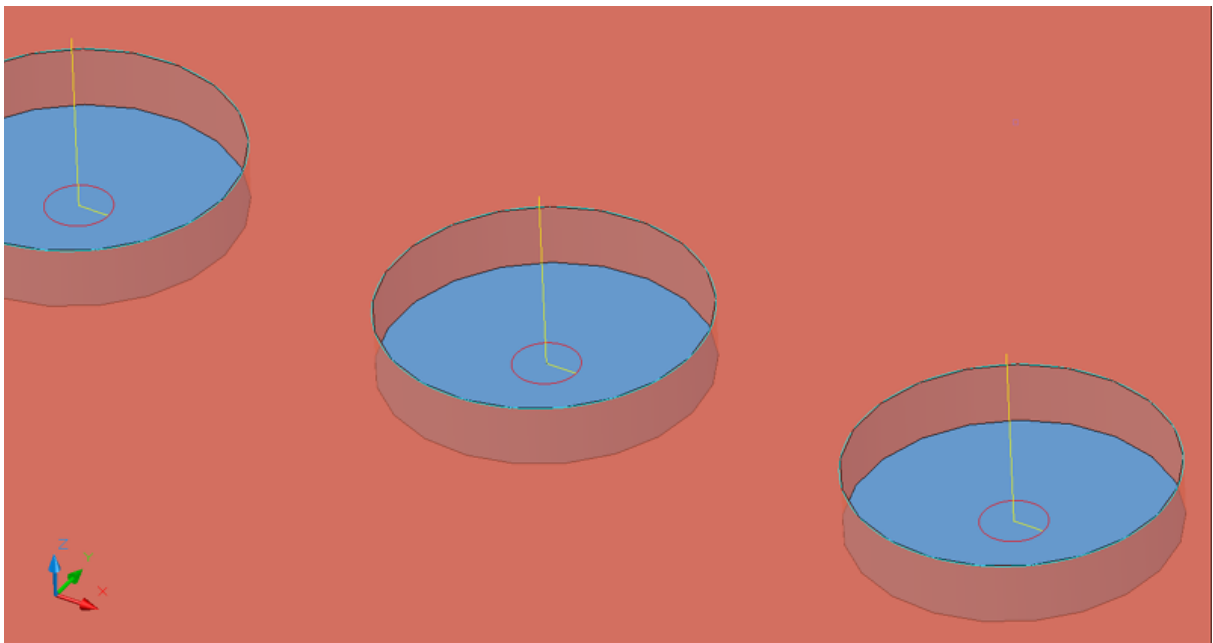
Hole-Interpolation 1



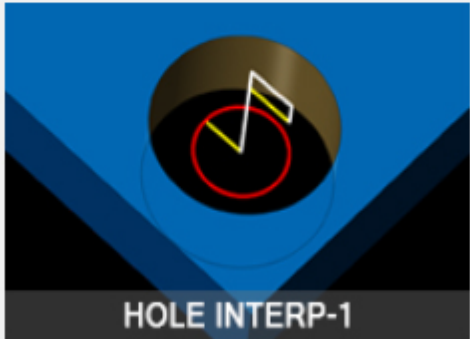
These cut cycles will allow the tool to start at the center of a circle, plunge down to the cut depth, interpolate the circle back to the starting point and then move back to the center of the circle and retract to the safety plane. The difference between them is that Hole Interpolation 2 will have an overlap in the cut past the start point before the tool moves back to the center of the circle.

This cycle only works on circles or arcs.

You should only use this cycle when there will be no slug left behind in the hole that could be expelled from the part and cause injury.



Hole-Interpolation 1 tool paths.

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▼	Safety Plane	*0.25000
Lead Feed	▼	Depth Per Pass	1.00000
Overlap Amount	AUTO ▼	Total Cut Depth	
XY Stock Allowance	▼	Feedrate/Spindle Speed	
Z Stock Allow	▼	Feedrate	350.00000
		Spindle Speed	18000.00000
		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Hole-Interpolation-1 Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the [Offset Dim](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See [Z Stock Allowance](#) for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

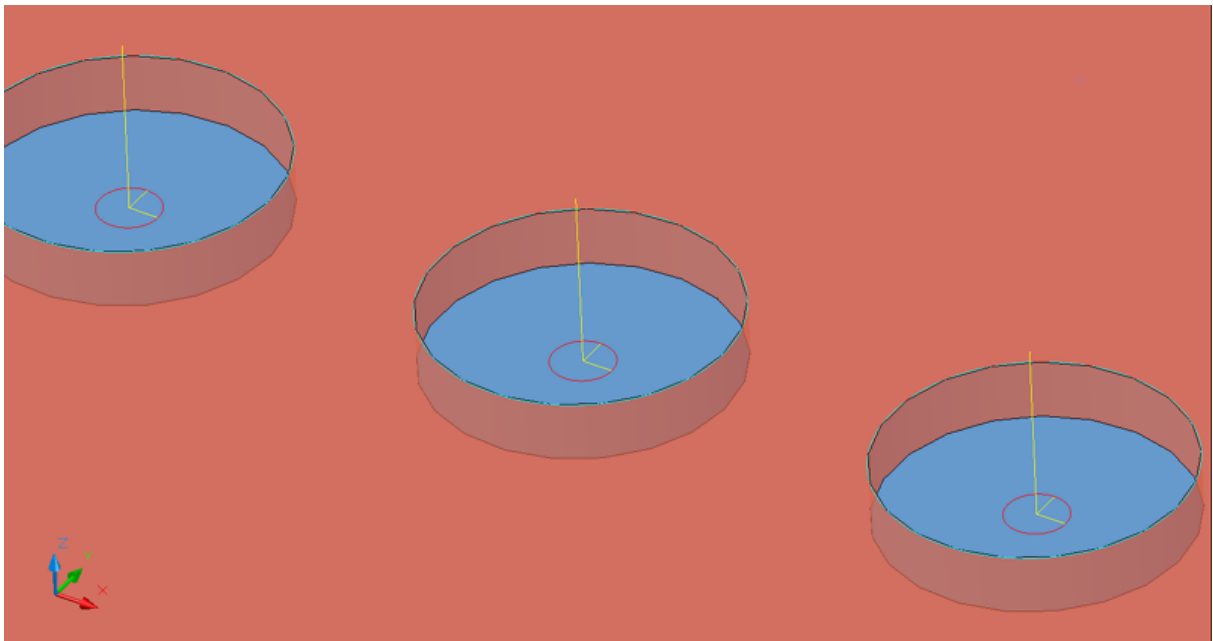
Hole-Interpolation 2



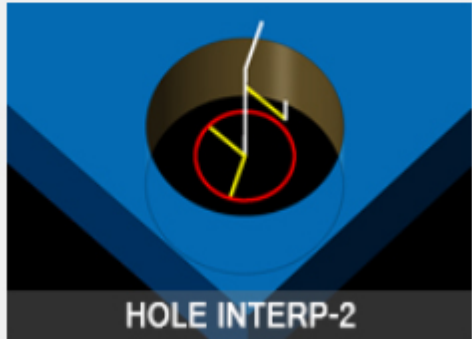
These cut cycles will allow the tool to start at the center of a circle, plunge down to the cut depth, interpolate the circle back to the starting point and then move back to the center of the circle and retract to the safety plane. The difference between them is that Hole Interpolation 2 will have an overlap in the cut past the start point before the tool moves back to the center of the circle.

This cycle only works on circles or arcs.

You should only use this cycle when there will be no slug left behind in the hole that could be expelled from the part and cause injury.



Hole-Interpolation 2 tool paths.

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▼	Safety Plane	*0.25000
Lead Feed	▼	Depth Per Pass	1.00000
Overlap Amount	AUTO ▼	Total Cut Depth	
XY Stock Allowance	▼	Feedrate/Spindle Speed	
Z Stock Allow	▼	Feedrate	350.00000
		Spindle Speed	18000.00000
		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Hole-Interpolation 2 Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (FIRSTXY XYCUTLOC) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the [Offset Dim](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See [Z Stock Allowance](#) for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Pocketing Cycles

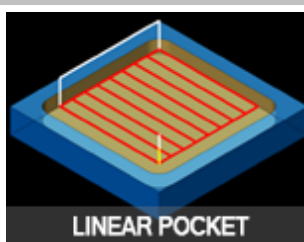
In pocketing, you have to remove material from the interior of a closed geometry. Because there are many types of tools and materials, there are several pocketing cycles in Router-CIM. Each is intended to behave slightly differently to accomplish the task of pocketing with various results. Some of the cycles will make a linear motion to clear the pocket, and some will make pattern following motions to clear the pocket. Each cycle has the ability to make clean up passes around the perimeter of the shape and also separate passes to clean up around islands that may be in the shape.

When the geometry consists of one or more profiles and none of them are enclosed or intersect with one another, each is cut as a separate pocket without islands.

When the geometry consists of one or more profiles, any of them which are enclosed or intersect with another are treated as an island. You can define several islands within a single pocket.

The descriptions given for each cycle will explain the various parameters available and how they function.

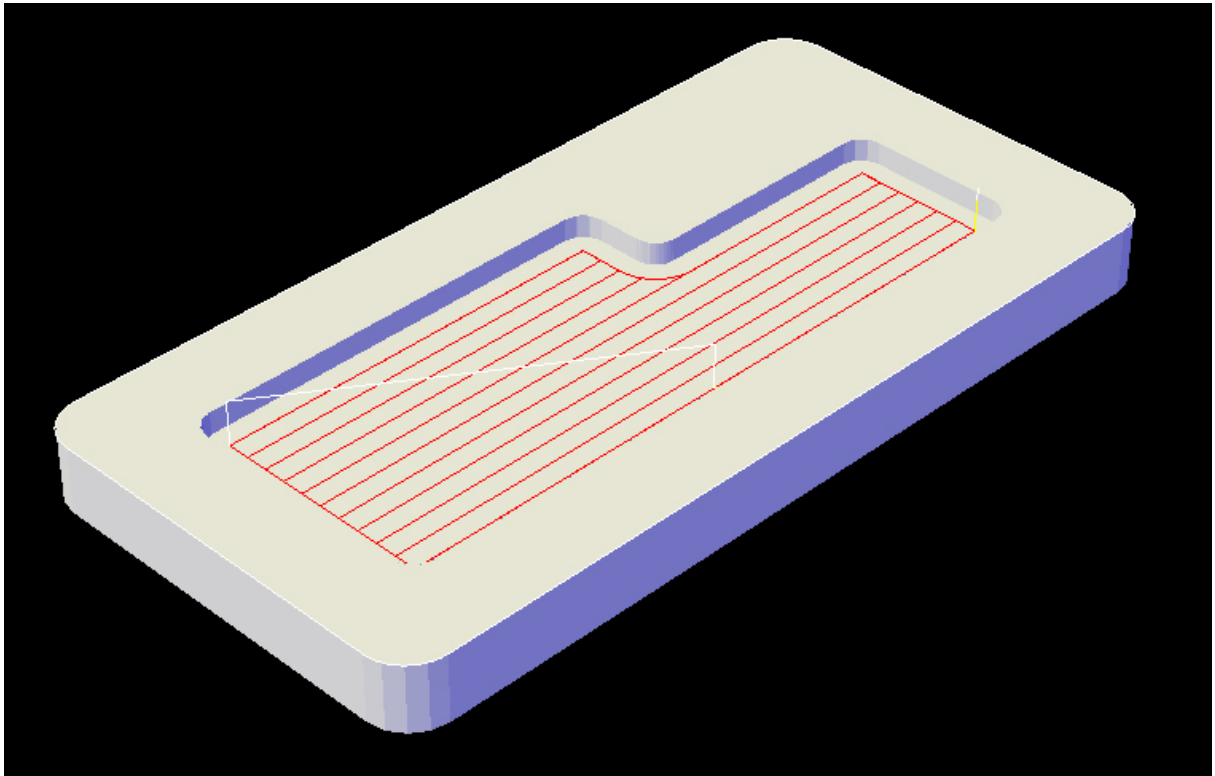
Linear-Pocket



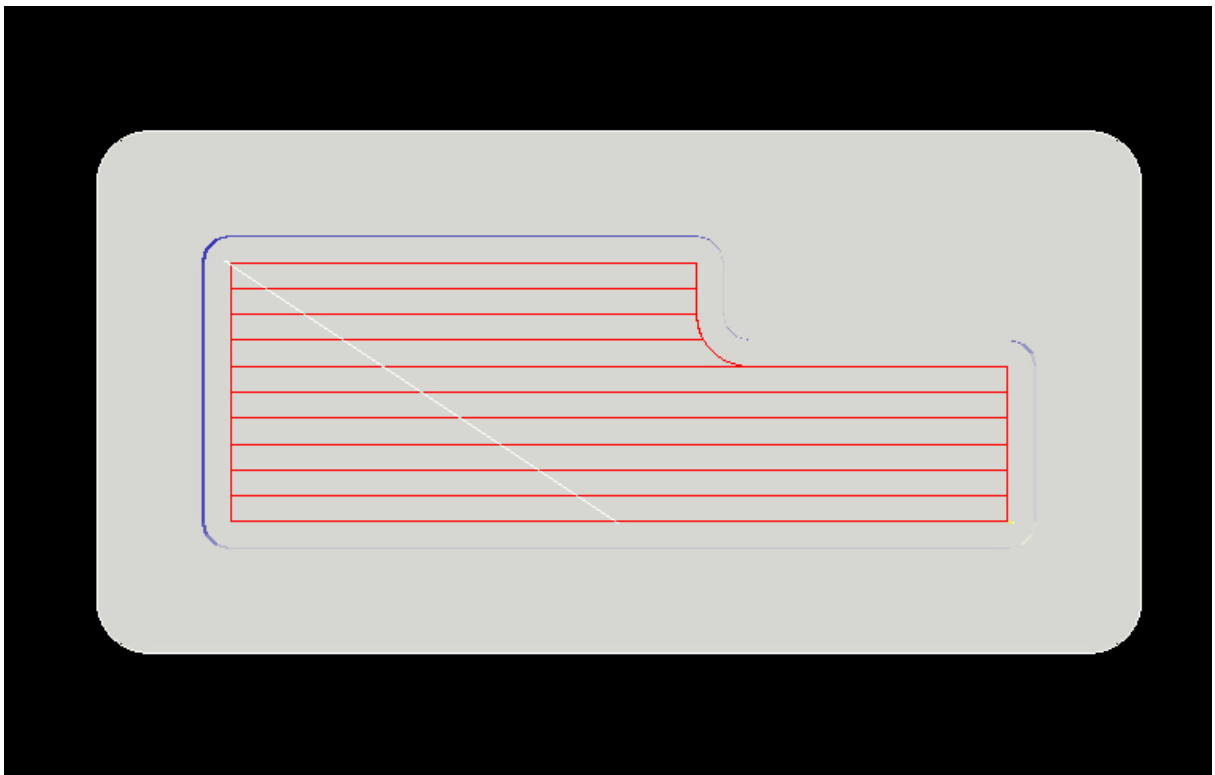
Linear Pocketing will move the tool across the shape to be cut in a straight back and forth pattern, removing material with each pass. There is a provision for a finish pass to clean up the profile of the shape and also provisions for clean up passes around islands in the shape.

The direction of the cut (in degrees) can be controlled give a cut in the X direction, Y direction, or at any angle.

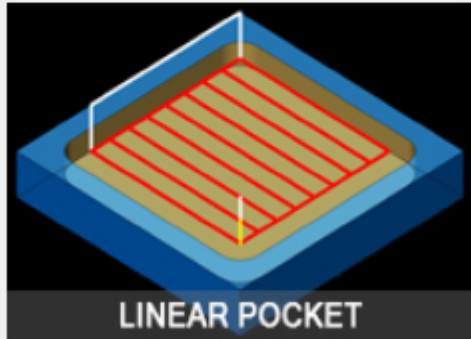
Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.



Linear Pocket Tool Paths



Linear Pocket Tool Paths

Cycle Information		Status Information	
Finish Pass	!*tr*	Safety Plane	*0.25000
Finish Allowance	0.001	Depth Per Pass	1.00000
Island Finish	N	Total Cut Depth	
Cut Spacing	!*tr*	Feedrate/Spindle Speed	
Cut Angle	N	Feedrate	350.00000
Cut Direction		Spindle Speed	18000.00000
Standoff Pass		Surface FPM	NONE
Collision Check		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
		 LINEAR POCKET	
		Reset Cycle Settings to Default	

Linear Pocket Parameters

The following parameters effect the toolpath creation:

Finish Pass

The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.

The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish

Either Y or N are valid values here.

N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.

If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing

This value is the spacing between each pass of the tool in the pocket.

Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass spacing to that number.

Cut Angle

Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut. The value given is in degrees.

N tells Router-CIM to determine the direction on its own.

Cut Direction

Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass

A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.

A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

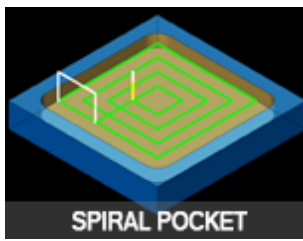
Collision Check

The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.

When set to Y, the pocketing routine will run slower than normal.

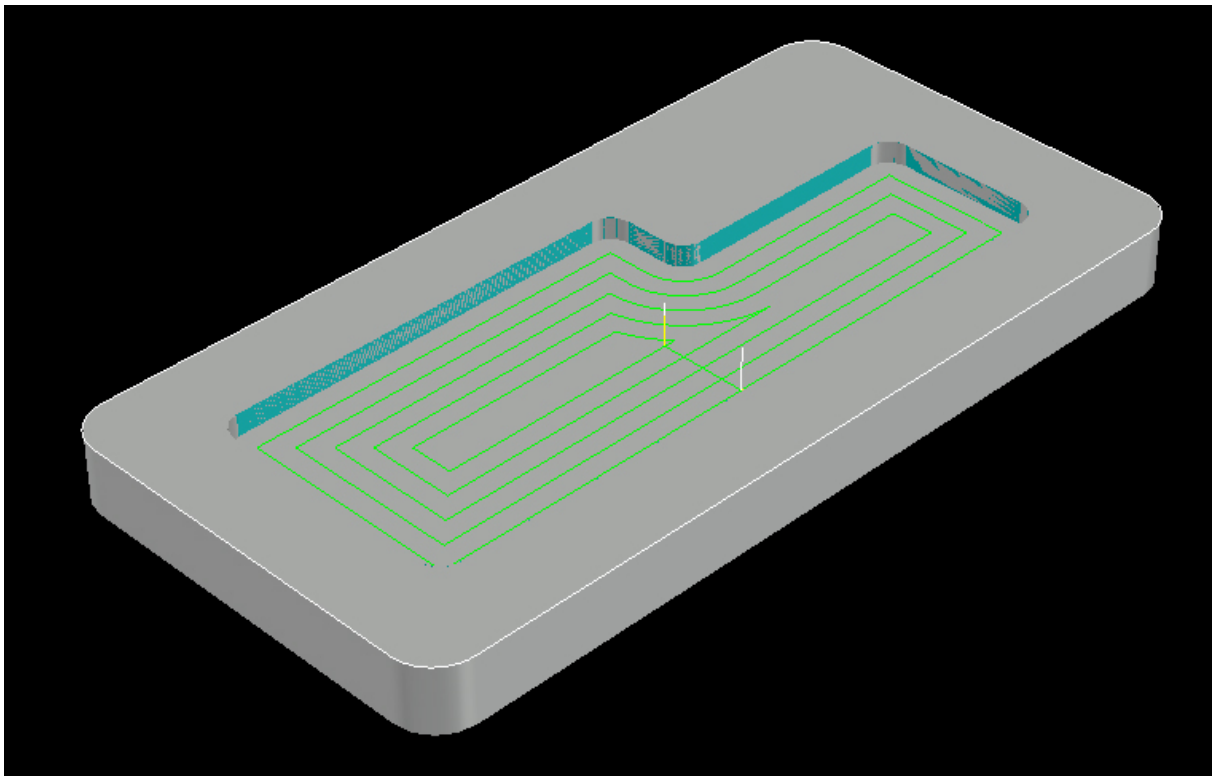
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Spiral-Pocket

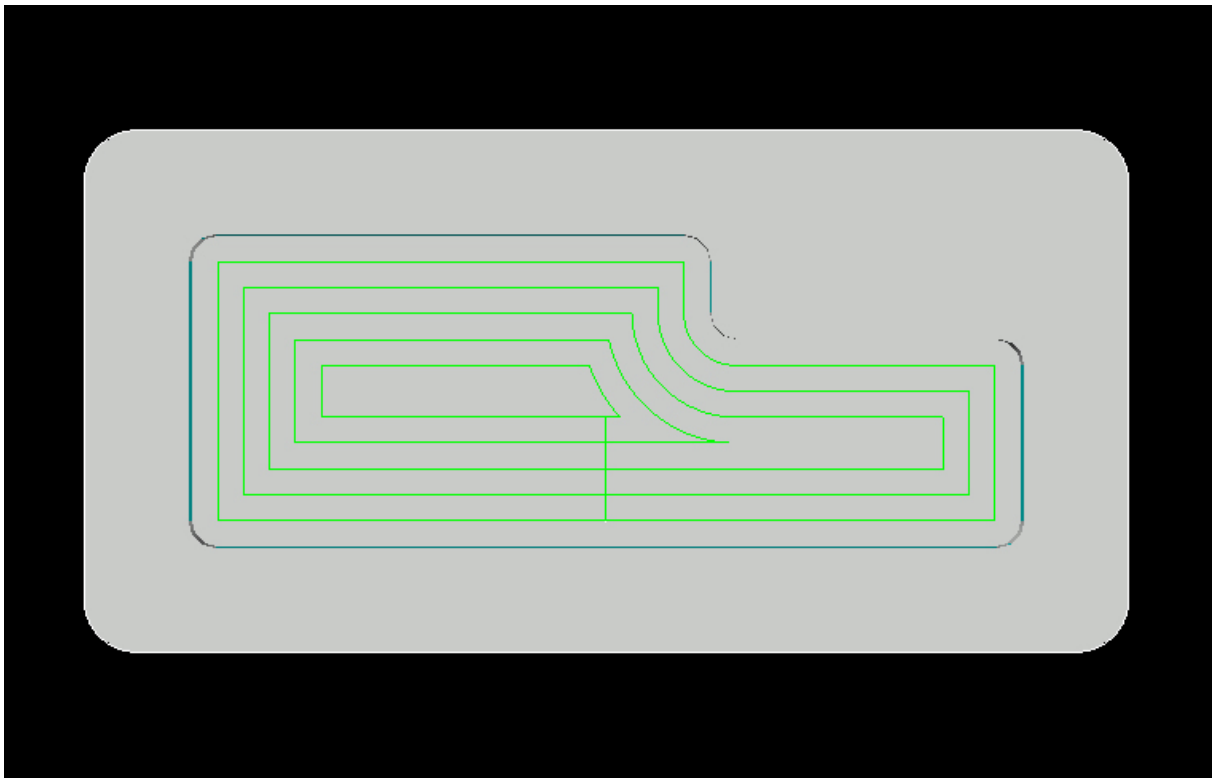


Spiral Pocketing will create pocketing tool motions that will be offsets of the shape of the pocket. A Spiral Pocket can contain islands and there are parameters in the cycle to allow for a finish pass around islands and also around the perimeter of the pocket.

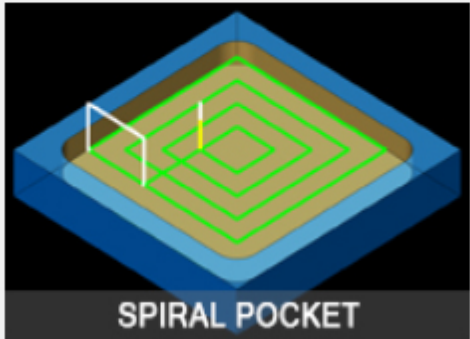
Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.



Spiral Pocket Tool Path



Spiral Pocket Tool Path

Cycle Information		Status Information	
Finish Pass	!*tr*	Safety Plane	*0.25000
Finish Allowance	0.001	Depth Per Pass	1.00000
Island Finish	N	Total Cut Depth	
Cut Spacing	!*tr*	Feedrate/Spindle Speed	
Cut Angle	N	Feedrate	350.00000
Cut Direction	CW	Spindle Speed	18000.00000
Standoff Pass		Surface FPM	NONE
Collision Check		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Spiral Pocketing Parameters.

The following parameters effect the toolpath creation:

Finish Pass

The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.

The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish

Either Y or N are valid values here.

N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.

If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing

This value is the spacing between each pass of the tool in the pocket.

Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass spacing to that number.

Cut Angle

N tells Router-CIM to determine the direction on its own. This is the default for Spiral Pocketing where the angle of each pass is determined by the geometry of the shape.

Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut with Linear Pocketing. It has no effect for Spiral Pocketing.

Cut Direction

Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass

A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.

A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

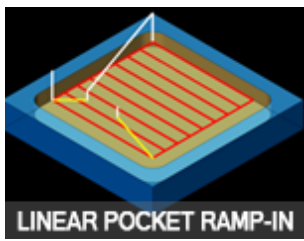
Collision Check

The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.

When set to Y, the pocketing routine will run slower than normal.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Linear Pocket Ramp-In

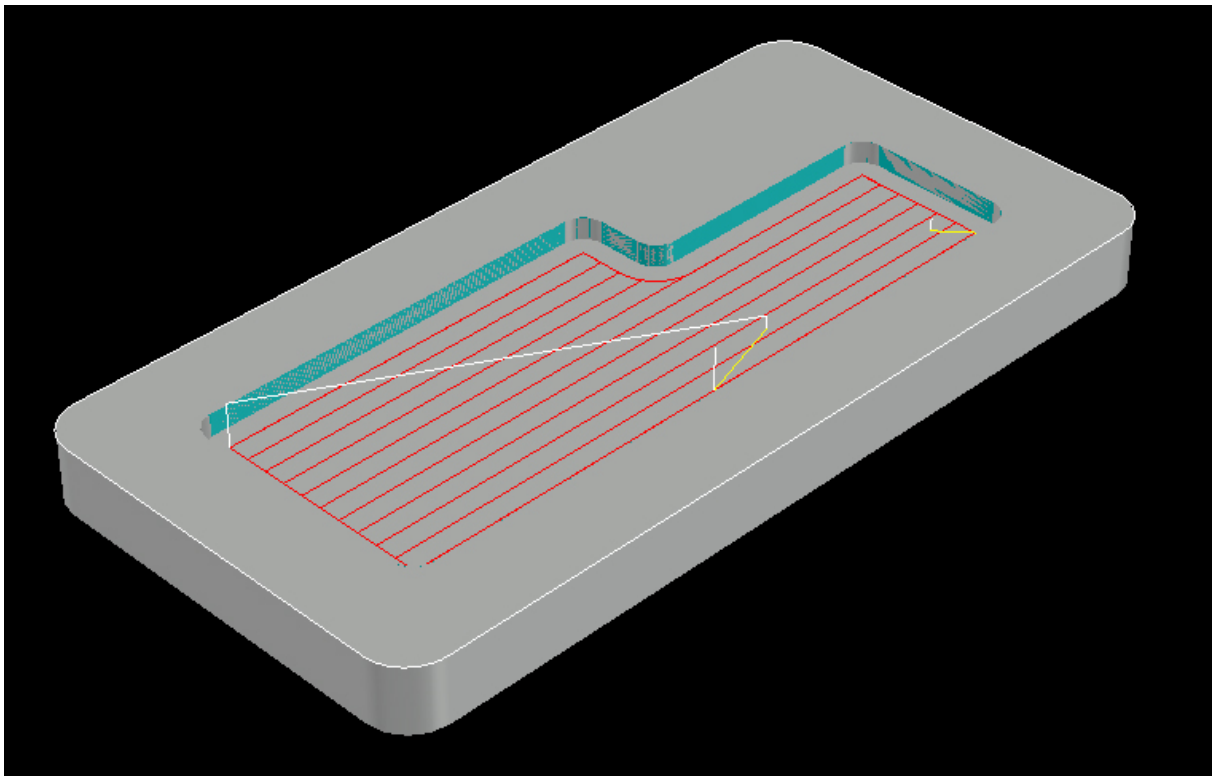


Linear Pocketing will move the tool across the shape to be cut in a straight back and forth pattern, removing material with each pass. There is a provision for a finish pass to clean up the profile of the shape and also provisions for clean up passes around islands in the shape.

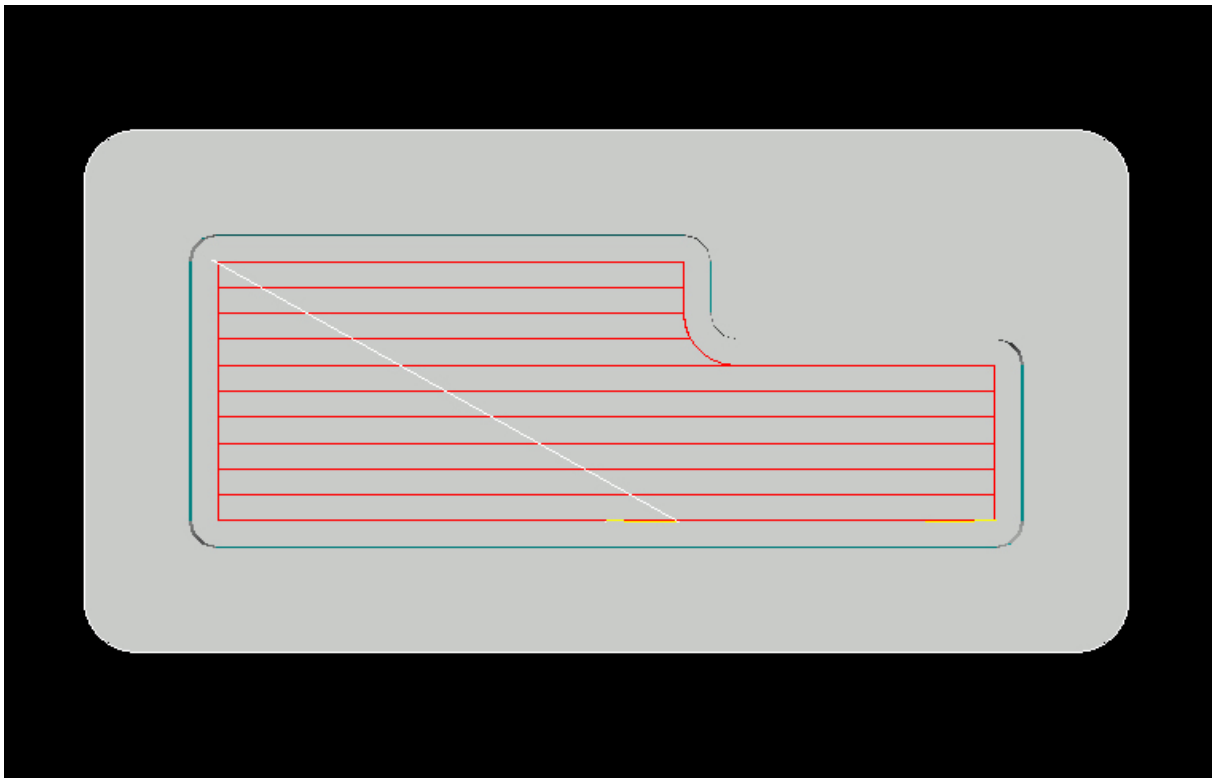
The difference with Linear Pocket Ramp-In is that the first pass will ramp into the material instead of making a plunge cut.

The direction of the cut (in degrees) can be controlled give a cut in the X direction, Y direction, or at any angle.

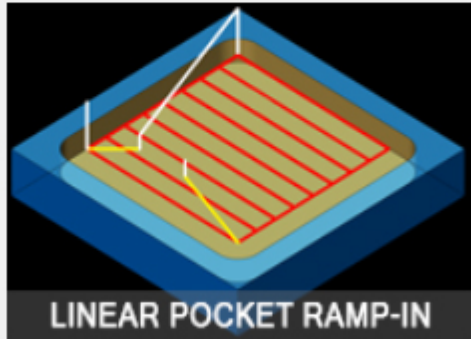
Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.



Linear Pocket Ramp-In Tool Path



Linear Pocket Ramp-In Tool Path

Cycle Information		Status Information	
Finish Pass	!*tr*	Safety Plane	*0.25000
Finish Allowance	0.001	Depth Per Pass	1.00000
Island Finish	N	Total Cut Depth	
Cut Spacing	!*tr*	Feedrate/Spindle Speed	
Cut Angle	N	Feedrate	350.00000
Cut Direction		Spindle Speed	18000.00000
Standoff Pass		Surface FPM	NONE
Collision Check		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Linear Pocket Ramp-In Parameters

The following parameters effect the toolpath creation:

Finish Pass

The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.

The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish

Either Y or N are valid values here.

N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.

If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing

This value is the spacing between each pass of the tool in the pocket.

Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass spacing to that number.

Cut Angle

Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut.

The value given is in degrees.

N tells Router-CIM to determine the direction on its own.

Cut Direction

Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass

A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.

A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

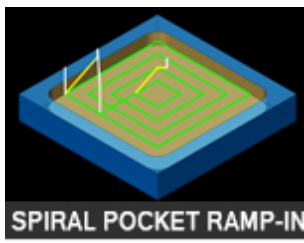
Collision Check

The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.

When set to Y, the pocketing routine will run slower than normal.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

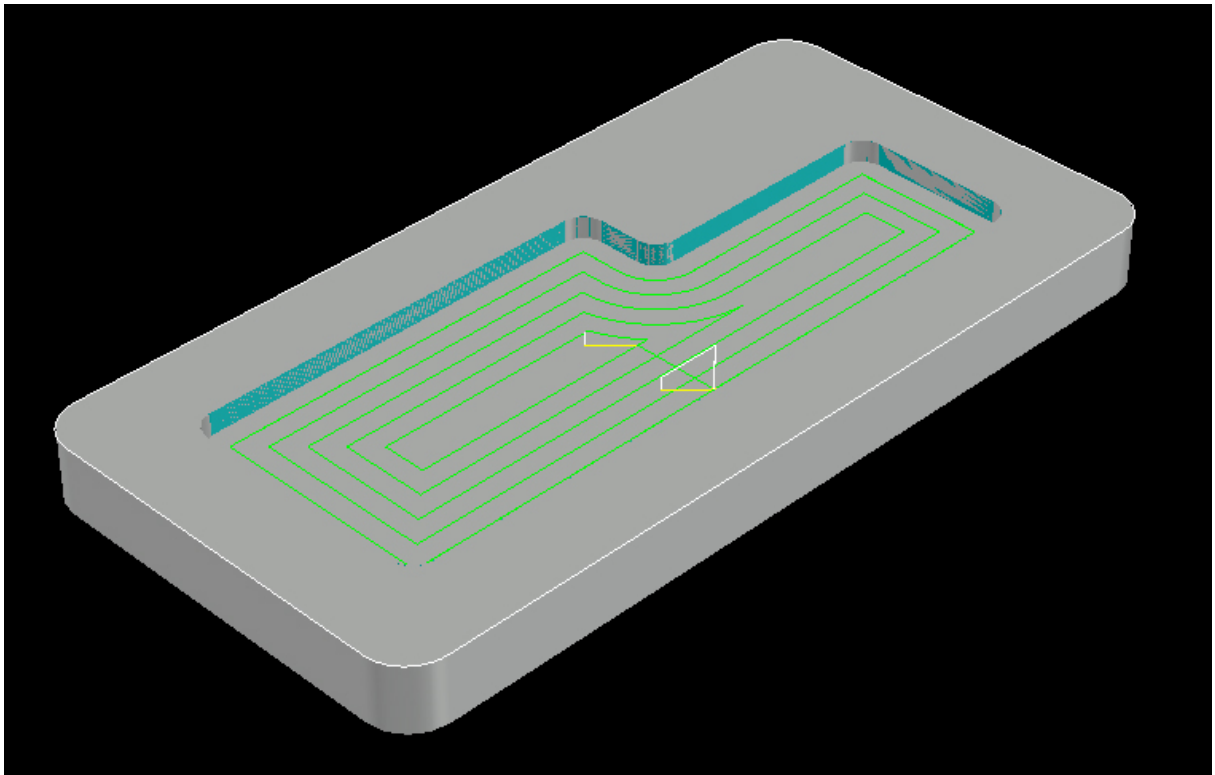
Spiral Pocket Ramp-In



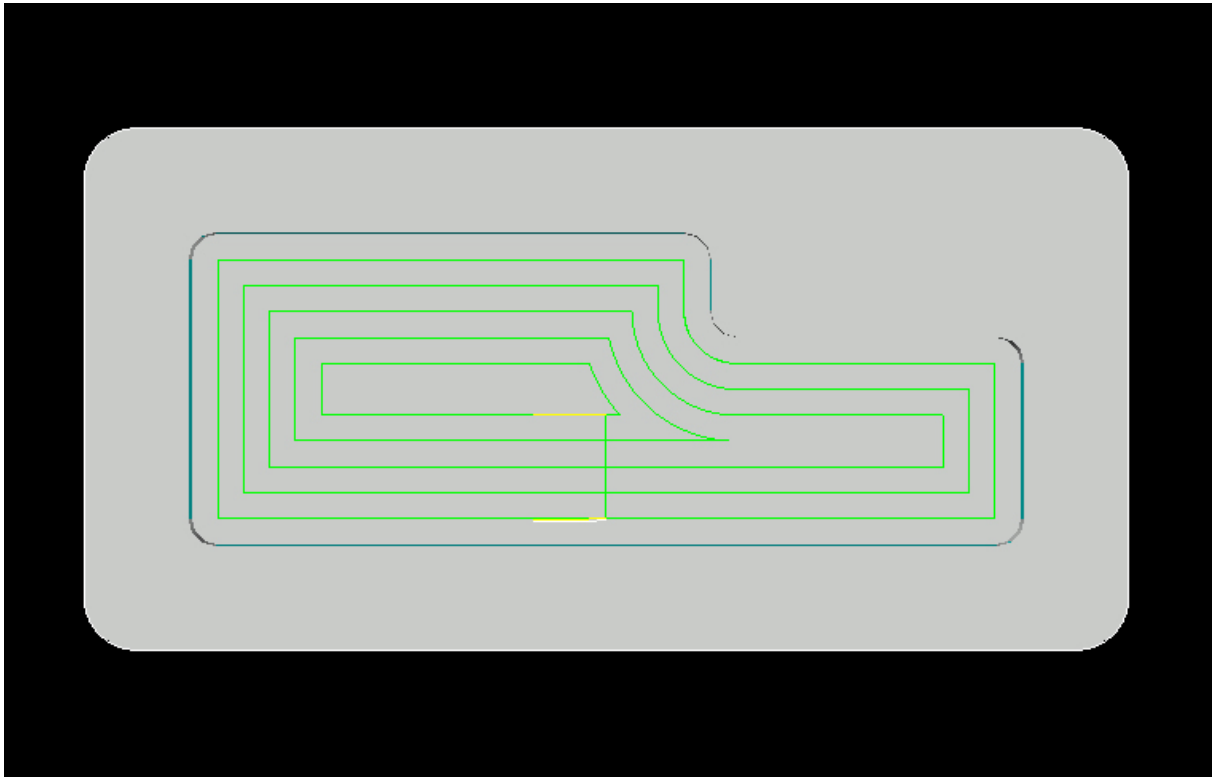
Spiral Pocketing will create pocketing tool motions that will be offsets of the shape of the pocket. A Spiral Pocket can contain islands and there are parameters in the cycle to allow for a finish pass around islands and also around the perimeter of the pocket.

The difference with Spiral Pocket Ramp-In is the move into the material will be a ramping lead in, instead of a plunge cut.

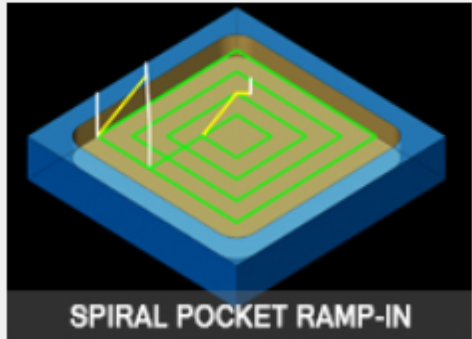
Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.



Spiral Pocket Ramp-In Tool Path



Spiral Pocket Ramp-In Tool Path

Cycle Information		Status Information	
Finish Pass	!*tr*	Safety Plane	*0.25000
Finish Allowance	0.001	Depth Per Pass	1.00000
Island Finish	N	Total Cut Depth	
Cut Spacing	!*tr*	Feedrate/Spindle Speed	
Cut Angle	N	Feedrate	350.00000
Cut Direction	CW	Spindle Speed	18000.00000
Standoff Pass		Surface FPM	NONE
Collision Check		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Spiral-Pocket-Ramp-In Parameters

The following parameters effect the toolpath creation:

Finish Pass

The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.

The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish

Either Y or N are valid values here.

N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.

If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing

This value is the spacing between each pass of the tool in the pocket.

Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass spacing to that number.

Cut Angle

N tells Router-CIM to determine the direction on its own. This is the default for Spiral Pocketing where the angle of each pass is determined by the geometry of the shape.

Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut with Linear Pocketing. It has no effect for Spiral Pocketing.

Cut Direction

Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass

A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.

A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

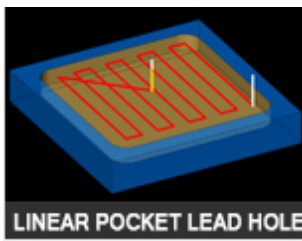
Collision Check

The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.

When set to Y, the pocketing routine will run slower than normal.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Linear Pocket Lead Hole



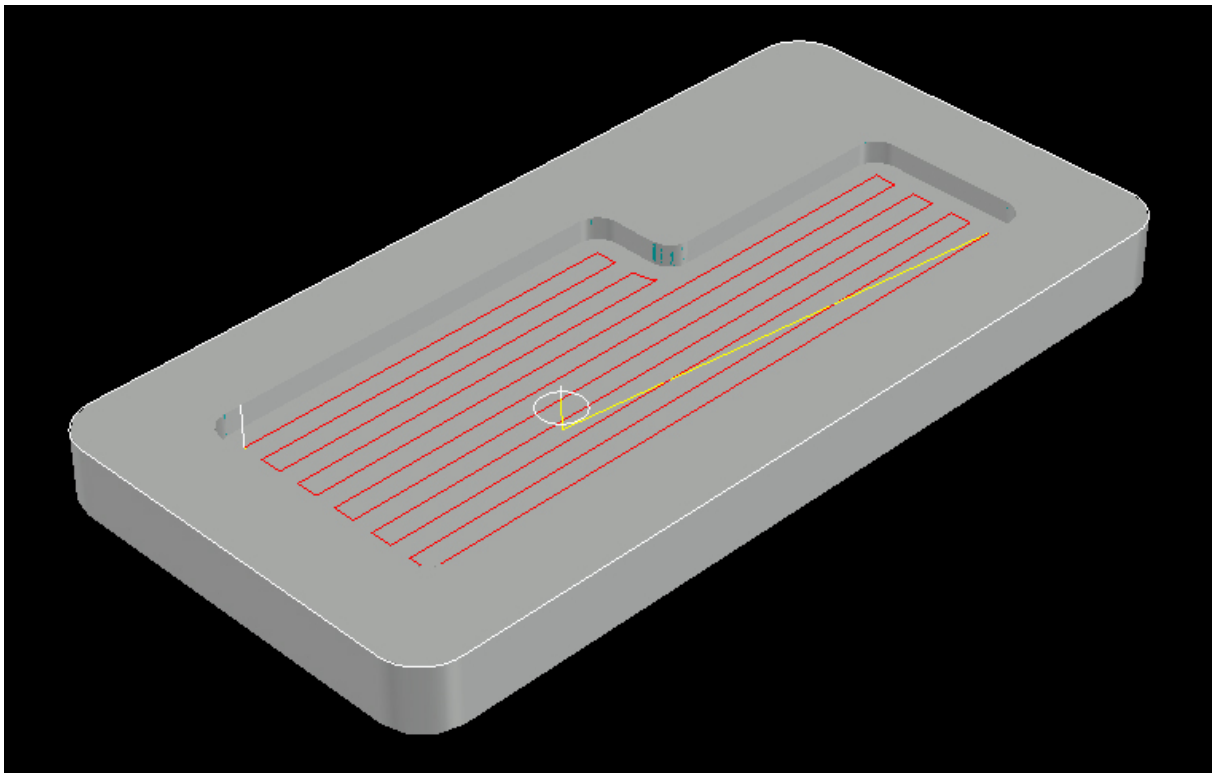
Linear Pocketing will move the tool across the shape to be cut in a straight back and forth pattern, removing material with each pass. There is a provision for a finish pass to clean up the profile of the shape and also provisions for clean up passes around islands in the shape.

The direction of the cut (in degrees) can be controlled give a cut in the X direction, Y direction, or at any angle.

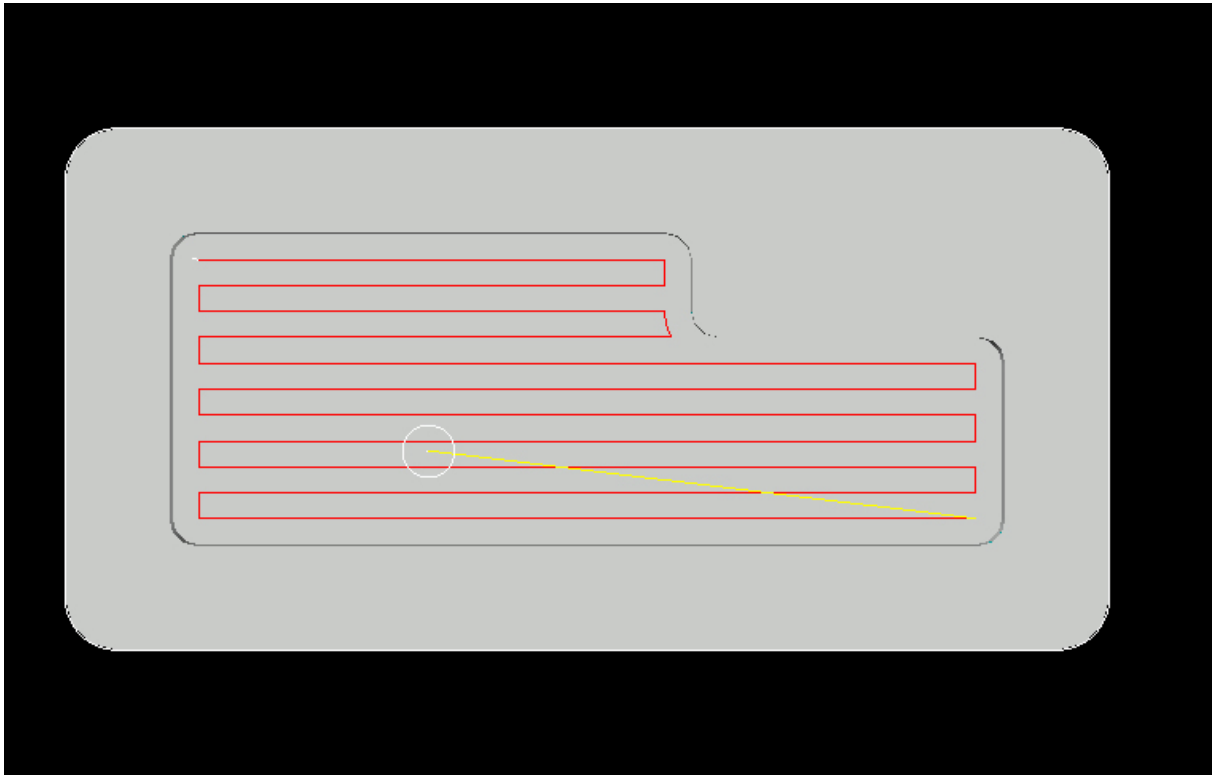
Linear Pocket Lead-Hole will allow the tool to plunge into a hole that has already been created in the part to avoid the tool plunging into the material and breaking.

Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.

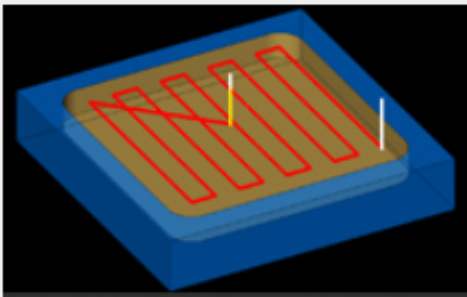
During the cut, you will be prompted to select the lead hole location. Use the center point OSNAP and pick the center of the hole into which you want the tool to plunge to make the lead in. Note that the tool will plunge into this hole and make a linear move at feedrate to the start of the shape, so the hole should be placed fairly close to the start point to avoid unnecessary cycle time.



Linear-Pocket-Lead-Hole Tool Path



Linear-Pocket-Lead-Hole Tool Path

Cycle Information		Status Information	
Finish Pass	!*tr*	Safety Plane	*0.25000
Finish Allowance	0.001	Depth Per Pass	1.00000
Island Finish	N	Total Cut Depth	
Cut Spacing	!*tr*	Feedrate/Spindle Speed	
Cut Angle	N	Feedrate	350.00000
Cut Direction		Spindle Speed	18000.00000
Standoff Pass		Surface FPM	NONE
Collision Check		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
		 LINEAR POCKET LEAD HOLE	
		Reset Cycle Settings to Default	

Linear Pocket Lead-Hole Parameters

The following parameters effect the toolpath creation:

Finish Pass

The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.

The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish

Either Y or N are valid values here.

N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.

If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing

This value is the spacing between each pass of the tool in the pocket.

Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass spacing to that number.

Cut Angle

Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut.

The value given is in degrees.

N tells Router-CIM to determine the direction on its own.

Cut Direction

Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass

A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.

A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

Collision Check

The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.

When set to Y, the pocketing routine will run slower than normal.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

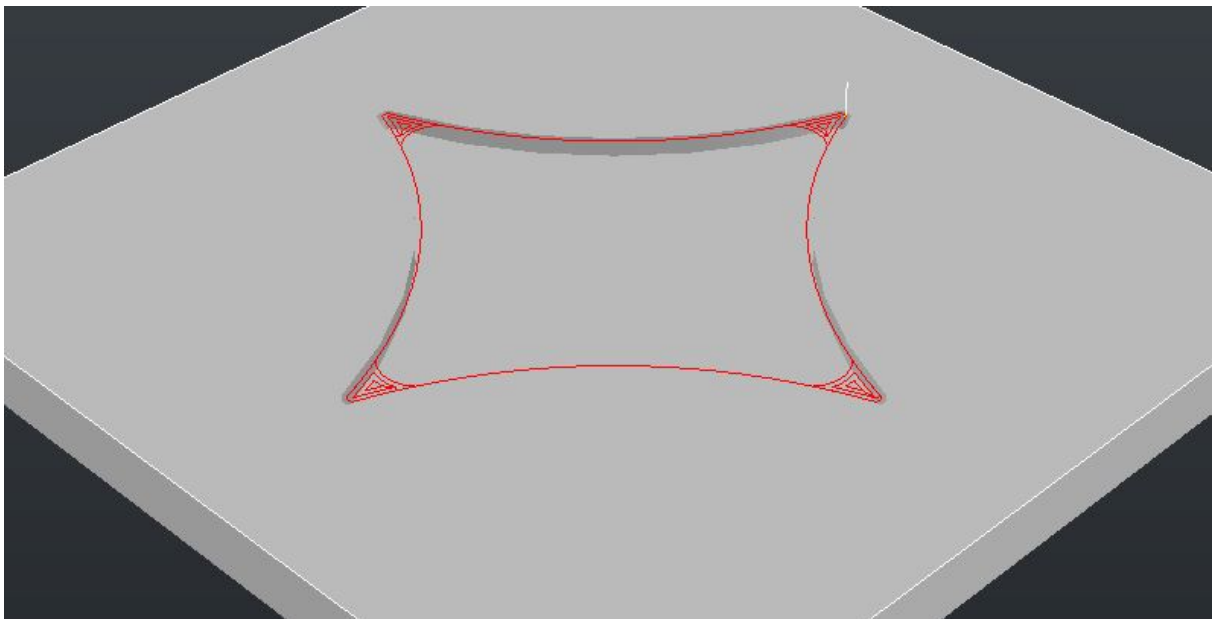
Advanced Rest Pocketing (Adv. Rest Pocketing)




This version of Advanced Pocketing allows you to use this cutting cycle in order to find the REST of the material left over from a previous pocketing cycle.

An example of this would be if you were to use a cutting tool that has a diameter larger than twice the diameter of the tool you may be using to apply a finish pass around a pocket. The Advanced Rest Pocketing would calculate the clean up needed in order to not leave material by adding additional cut paths.

Note: This cycle is only available on 64-bit Operating Systems. If you are running a 32-bit Operating System, you will be unable to use this cycle.



Advanced Rest Pocketing Tool Path

Cycle Information		Status Information	
Spiral-In	N	Safety Plane	*0.25000
Previous Radius	0.375	Depth Per Pass	1.00000
Finish Allowance	0.00	Total Cut Depth	
Step Over %	50	Feedrate/Spindle Speed	
Climb Mill	Y	Feedrate	350.00000
Clean Up Pass	N	Spindle Speed	18000.00000
Ramp In	NONE	Surface FPM	NONE
Ramp Angle	NONE	Units Per Revolution	NONE
Stay Down	Y	Calculate	
Save Shape	N	Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

The following parameters effect the toolpath creation:

Spiral-IN <N>

The value is not applicable to a Advanced Rest Pocketing function.

Rest Cutting <Y>

The value of 'Y' initiates the Advanced Rest Pocketing cycle.

Prev. Radius <0.0>

The value entered in this parameter is what the Advanced Rest Pocketing cycles uses to determine the amount of clean out passes it will need to make based on the tool that is selected for use in this cut cycle and what is entered into this parameter.

The value should be the radius of the tool that was used to pocket the overall geometry.

Finish Allow

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Step Over %

This value is the percentage of the tool diameter between each pass of the tool in the pocket. This needs to be a real number such as 25.0 or 50.0.

Climb Mill <Y>

The value represents if the cutting cycle will be doing Climb (CCW) milling <Y> or if you want the cutting cycle to do Conventional (CW) milling <N>

CleanUp Pass <N>

A CleanUp Pass is described as an additional tool path that travels around geometry allowing you to use Cutter Compensation for the boundary of the geometry.

Ramp-IN <N>

A Ramp-In set to <Y> will allow you to have the cutting cycle enter the cut with a ramp instead of a plunge. If this parameter is changed from the default of <N>, then the parameter of Ramp Angle will need to be defined in degrees.

Ramp Angle

If Ramp-IN is set to <Y>, then this parameter would need to be defined. The parameter will need a numeric value defining the degrees of the ramp such as 30 or 45.

Stay Down <N>

If this parameter is set to <Y>, it will keep the tool down while in the pocket to continue the shape. If it is set to <N>, it will allow the tool to pick up and move to another area of the pocket to continue the shape.

Save Shape

If this parameter is set to <Y>, it will apply the cut cycle as usual but it will also give you the geometry that was used to create the cut cycle. The geometry will be added to the layer 'NC_Shape' for additional tool paths if needed.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

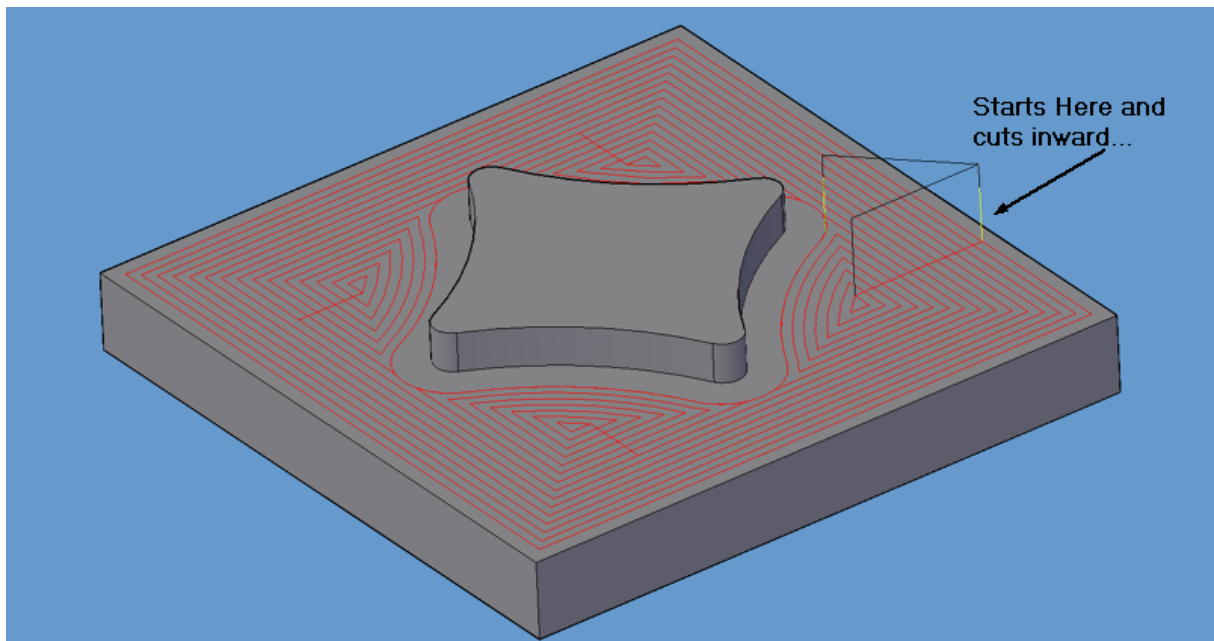
Advanced Inward Pocketing (Adv. Inward Pocketing)




This version of Advanced Pocketing uses a different algorithm and contains many options to control the creation of the tool path than the other pocketing cycles. This method allows for more complex geometry to be handed to the tool path generator.

The Advanced Inward Pocketing will start at a point at the outside edge of the pocket and create offset tool paths, moving toward the inside of the pocket. There are parameters to allow a plunge start or a ramp start, and also options for keeping the tool down while in the pocket or allowing the tool to pick up and move to another area of the pocket to continue the shape.

Note: This cycle is only available on 64-bit Operating Systems. If you are running a 32-bit Operating System, you will be unable to use this cycle.



Advanced Inward Tool Path

Cycle Information		Status Information	
Spiral-In	<input type="text" value="Y"/>	Safety Plane	<input type="text" value="0.25000"/>
Finish Allowance	<input type="text" value="0.00"/>	Depth Per Pass	<input type="text" value="1.00000"/>
Step Over %	<input type="text" value="50"/>	Total Cut Depth	<input type="text"/>
Climb Mill	<input type="text" value="Y"/>	Feedrate/Spindle Speed	
Clean Up Pass	<input type="text" value="N"/>	Feedrate	<input type="text" value="350.00000"/>
Ramp In	<input type="text" value="NONE"/>	Spindle Speed	<input type="text" value="18000.00000"/>
Ramp Angle	<input type="text" value="NONE"/>	Surface FPM	<input type="text" value="NONE"/>
Stay Down	<input type="text" value="Y"/>	Units Per Revolution	<input type="text" value="NONE"/>
Save Shape	<input type="text" value="N"/>	<input type="button" value="Calculate"/>	
		Before Codes	<input type="text"/>
		After Codes	<input type="text"/>
		Oscillation Amount	<input type="text" value="0.00000"/>
		Sort By Rank #	<input type="text"/>
			
		<input type="button" value="Reset Cycle Settings to Default"/>	

The following parameters effect the toolpath creation:

Spiral-IN <Y>

The value of 'Y' defines that the cycle will start on the outside portion of the geometry and work its way to the inside of the geometry. To have it work from the inside of the geometry to the outside, select the cutting cycle [Advanced Outward Pocketing](#).

Finish Allow

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Step Over %

This value is the percentage of the tool diameter between each pass of the tool in the pocket. This needs to be a real number such as 25.0 or 50.0.

Climb Mill <Y>

The value represents if the cutting cycle will be doing Climb (CCW) milling <Y> or if you want the cutting cycle to do Conventional (CW) milling <N>

CleanUp Pass <N>

A CleanUp Pass is described as an additional tool path that travels around geometry allowing you to use Cutter Compensation for the boundary of the geometry.

Ramp-IN <N>

A Ramp-In set to <Y> will allow you to have the cutting cycle enter the cut with a ramp instead of a plunge. If this parameter is changed from the default of <N>, then the parameter of Ramp Angle will need to be defined in degrees.

Ramp Angle

If Ramp-IN is set to <Y>, then this parameter would need to be defined. The parameter will need a numeric value defining the degrees of the ramp such as 30 or 45.

Stay Down <N>

If this parameter is set to <Y>, it will keep the tool down while in the pocket to continue the shape. If it is set to <N>, it will keep allow the tool to pick up and move to another area of the pocket to continue the shape.

Save Shape

If this parameter is set to <Y>, it will apply the cut cycle as usual but it will also give you the geometry that was used to create the cut cycle. The geometry will be added to the layer 'NC_Shape' for additional tool paths if needed.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

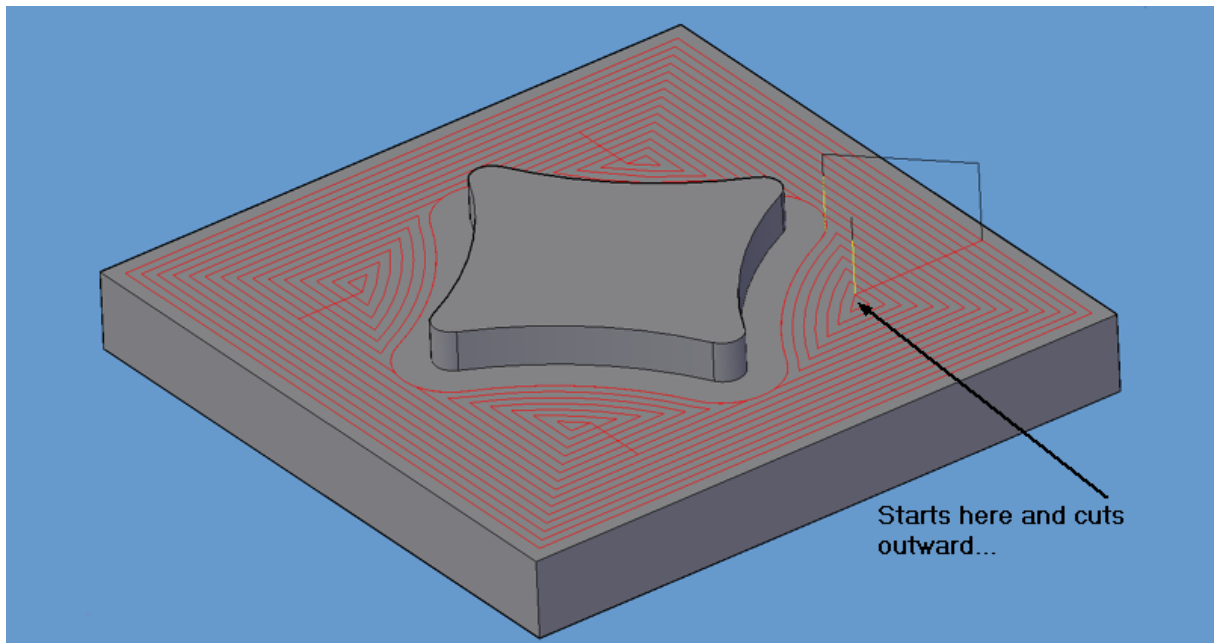
Advanced Outward Pocketing (Adv. Outward Pocketing)




This version of Advanced Pocketing uses a different algorithm and contains many options to control the creation of the tool path than the other pocketing cycles. This method allows for more complex geometry to be handed to the tool path generator.

The Advanced Outward Pocketing will start at a point inside the pocket and create offset tool paths, moving toward the outside of the pocket. There are parameters to allow a plunge start or a ramp start, and also options for keeping the tool down while in the pocket or allowing the tool to pick up and move to another area of the pocket to continue the shape.

Note: This cycle is only available on 64-bit Operating Systems. If you are running a 32-bit Operating System, you will be unable to use this cycle.



Advanced Outward Tool Path

Cycle Information		Status Information	
Spiral-In	N	Safety Plane	*0.25000
Finish Allowance	0.00	Depth Per Pass	1.00000
Step Over %	50	Total Cut Depth	
Climb Mill	Y	Feedrate/Spindle Speed	
Clean Up Pass	N	Feedrate	350.00000
Ramp In	NONE	Spindle Speed	18000.00000
Ramp Angle	NONE	Surface FPM	NONE
Stay Down	Y	Units Per Revolution	NONE
Save Shape	N	Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

The following parameters effect the toolpath creation:

Spiral-IN <N>

The value of 'N' defines that the cycle will start on the inside portion of the geometry and work its way to the outside of the geometry. To have it work from the outside of the geometry to the inside, select the cutting cycle ['Advanced Inward Pocketing'](#).

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Step Over %

This value is the percentage of the tool diameter between each pass of the tool in the pocket. This needs to be a real number such as 25.0 or 50.0.

Climb Mill <Y>

The value represents if the cutting cycle will be doing Climb (CCW) milling <Y> or if you want the cutting cycle to do Conventional (CW) milling <N>.

CleanUp Pass <N>

A CleanUp Pass is described as an additional tool path that travels around geometry allowing you to use Cutter Compensation for the boundary of the geometry.

Ramp-IN <N>

A Ramp-In set to <Y> will allow you to have the cutting cycle enter the cut with a ramp instead of a plunge. If this parameter is changed from the default of <N>, then the parameter of Ramp Angle will need to be defined in degrees.

Ramp Angle

If Ramp-IN is set to <Y>, then this parameter would need to be defined. The parameter will need a numeric value defining the degrees of the ramp such as 30 or 45.

Stay Down <N>

If this parameter is set to <Y>, it will keep the tool down while in the pocket to continue the shape. If it is set to <N>, it will keep allow the tool to pick up and move to another area of the pocket to continue the shape.

Save Shape

If this parameter is set to <Y>, it will apply the cut cycle as usual but it will also give you the geometry that was used to create the cut cycle. The geometry will be added to the layer 'NC_Shape' for additional tool paths if needed.

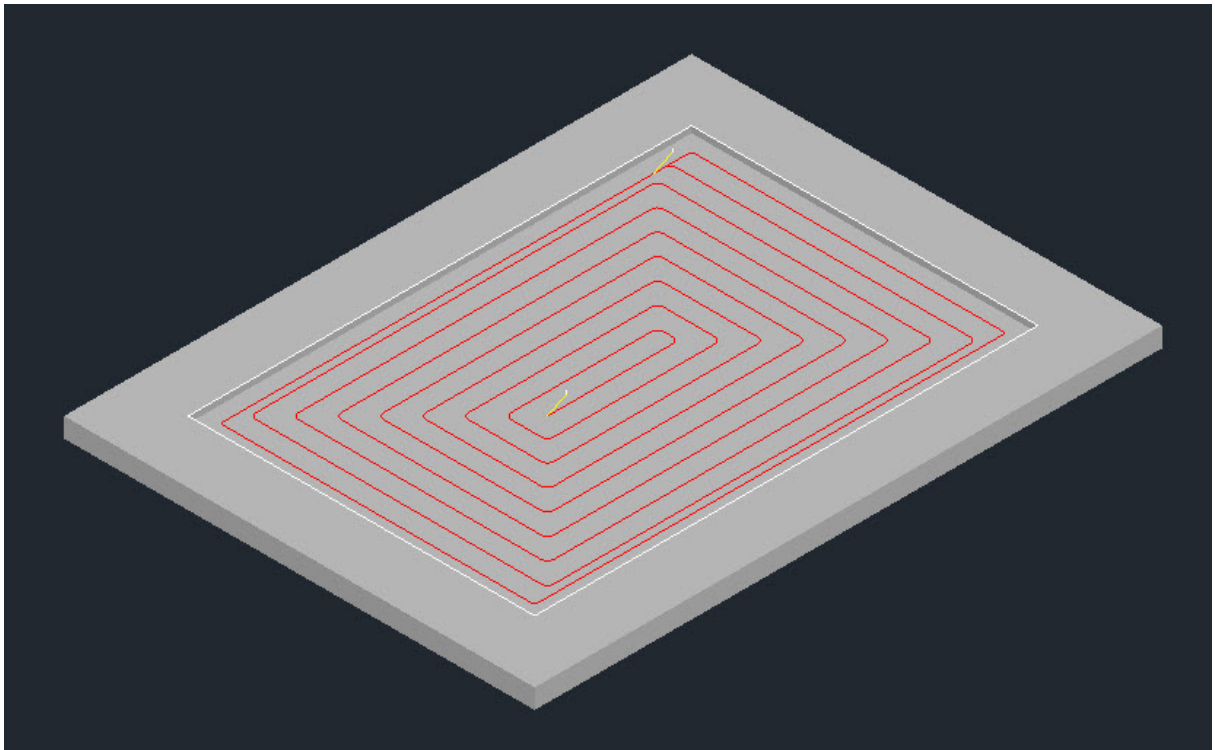
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

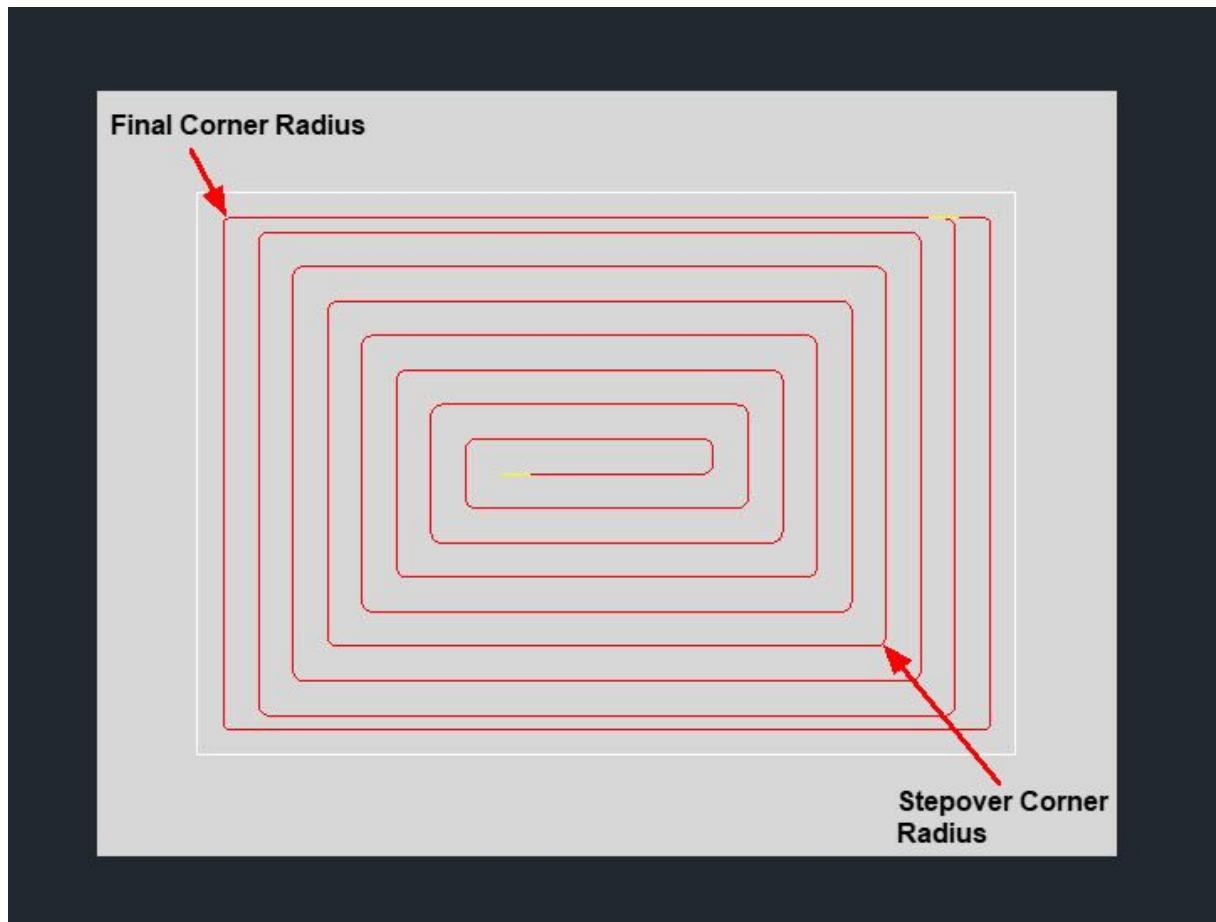
Rectangular Fast Pocket




The Rectangular Fast Pocket cycle gives you the ability to create a radius in the corner of the pocket to prevent the machine from coming to a stop when making a corner move. The Rectangular Fast Pocket cycle also allows for a unique corner radius for the final corners to make sure you are getting the corners you would expect. Take advantage of up to a 75% step over in the Rectangular Fast Pocket cycle.

Note: This cycle will only work with a rectangular shape.





Cycle Information		Status Information	
Offset Dim	0.0	Safety Plane	*0.25000
Cut Side	LH	Depth Per Pass	2.00000
Cut Direction	CW	Total Cut Depth	-0.25000
Round Corners	n	Feederate/Spindle Speed	
Lead In	N	Feederate	1000.00000
Lead Out	N	Spindle Speed	18000.00000
Lead Size	0.0	Surface FPM	NONE
Lead Ratio		Units Per Revolution	NONE
Lead Feed		Ramp Feederate	
XY Stock Allowance		Calculate	
Z Stock Allowance		Before Codes	
Finish Allowance		After Codes	
Stepover %		Oscillation Amount	0.00000
Stepover corner radius		Sort By Rank #	
Final corner radius		 RECTANGULAR FAST POCKET	
Final Overlap amount	*tr*		
		Reset Cycle Settings to Default	

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (0.0) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the [Offset Dim](#) section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Ratio

Lead Ratio determines the angle of the ramp in Z during the lead in and lead out. You can specify the Lead Ratio as a number that reflects the percentage of the angle from its default. That means that if you want a lead that is twice the normal ramp length (shallower angle) enter 2. If you want a lead that is steeper than the default, enter .5.

See the [Lead Ratio](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.

Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Stepover % (Required)

This value is the percentage of the tool diameter in decimal between each pass of the tool in the pocket. This needs to be a real number between 0.05 and 0.75.

Stepover corner radius (Required)

The value entered here will become an arc move for each corner that the pocketing pass does.

Final corner radius (Required)

The value entered here will become an arc move for each corner on the final pass of the pocket.

Final Overlap amount (Required)

Final Overlap amount is the movement of the cutter past the starting point of the cut. By default the overlap amount is equal to the radius of the tool (*tr*). You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the overlap distance is 0.25". If you put 1.0" in the overlap Amt. field then the overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the [Overlap Amt](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

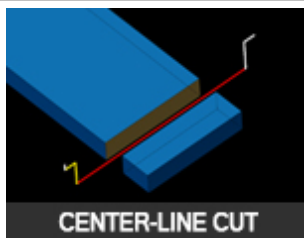
Open Shape and Center Cutting Cycles

There may be several instances where cutting an open shape is desirable. One instance may be to use a profile or shaper tool to cut a visible edge, leaving the profile shape in the material. In that instance you need a cycle where you can control the cut side and direction of the cut to perform either a climb or conventional cut.

Center cutting cycles are also in this section. There are many instances where cutting a shape on center with no offset is necessary, such as engraving or slotting. Router-CIM has several cycles available to allow complete control over the cutting conditions.

The descriptions given for each cycle will explain the various parameters available and how they function.

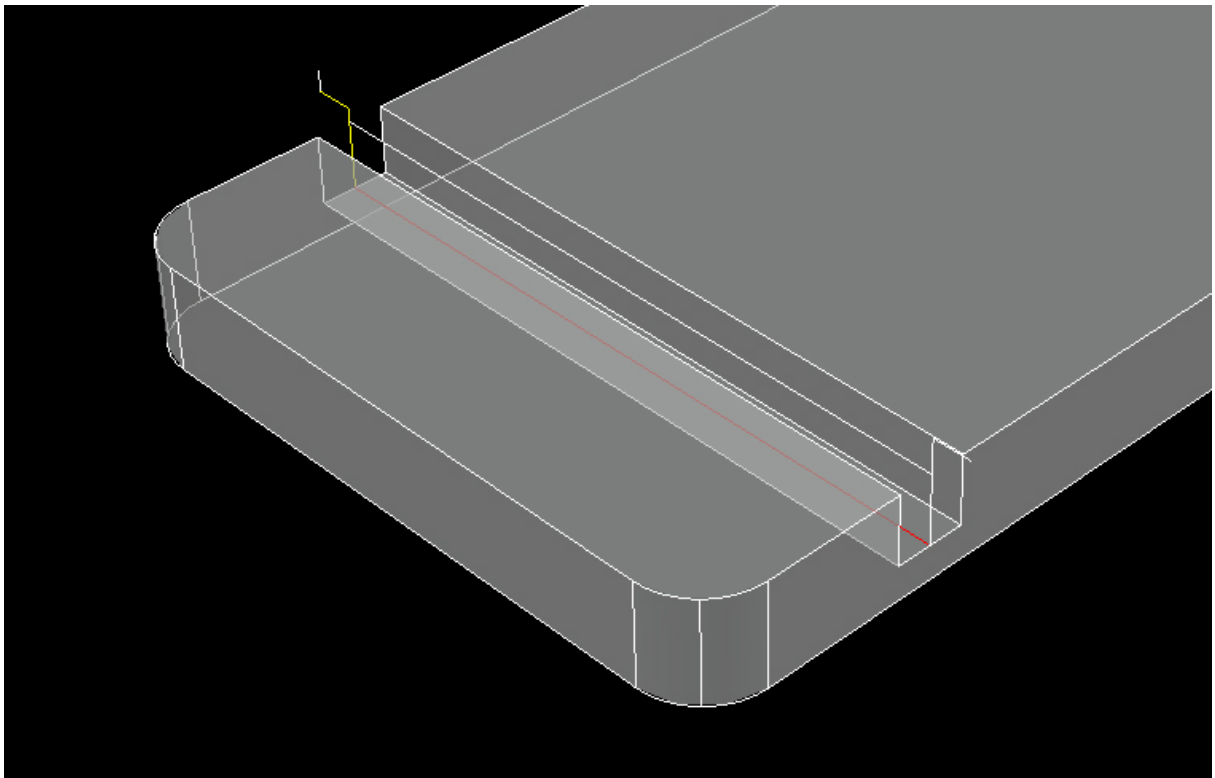
Center-Line Cut



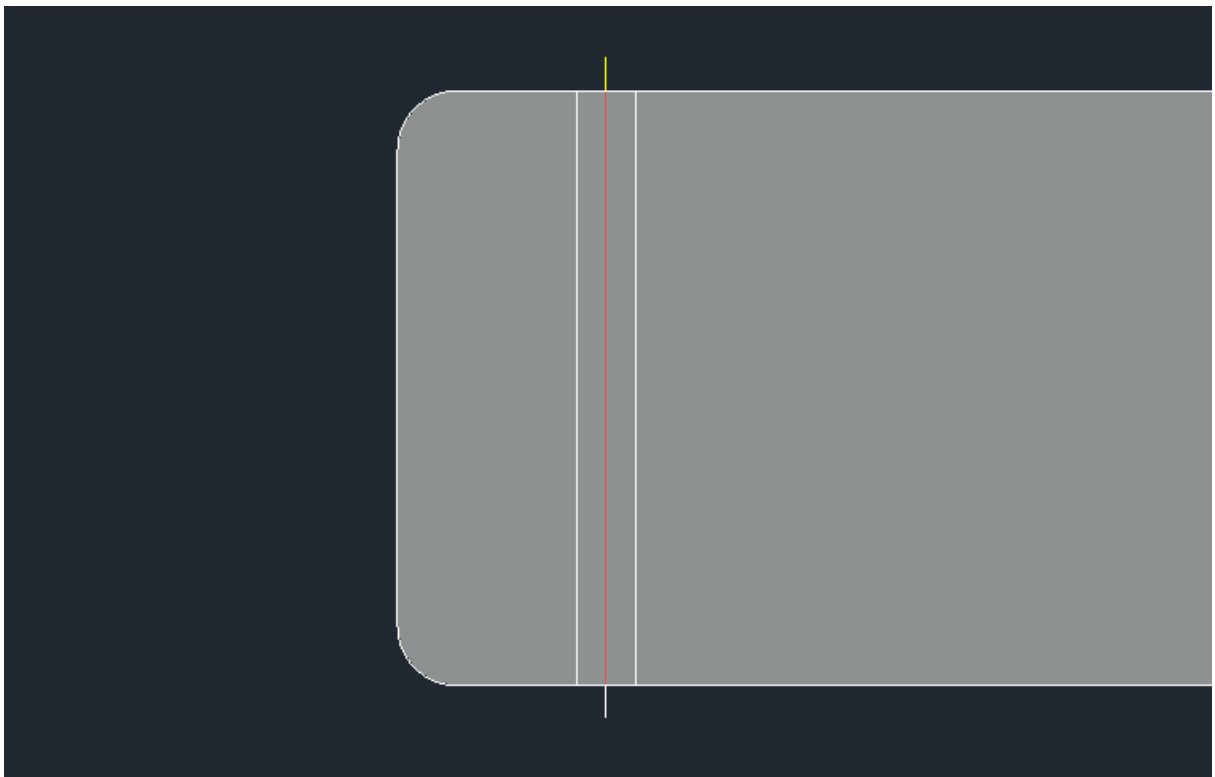
These cycles allow you to create tool paths directly from any defined shapes. Use this cycle to make a fixture or to engrave geometry. This cycle is also used to cut tool-width slots. You can draw the gasket grooves and vacuum grooves for a spoil board, and use this cycle to follow that geometry. This cycle would normally be used with a tool that does not use Machine Cutter Compensation, since there is no offset created.

Cycle Start and End positions determine where to rapid to and where to feed to in the Z axis. Tool Depth of Cut affects the number of cutting passes.

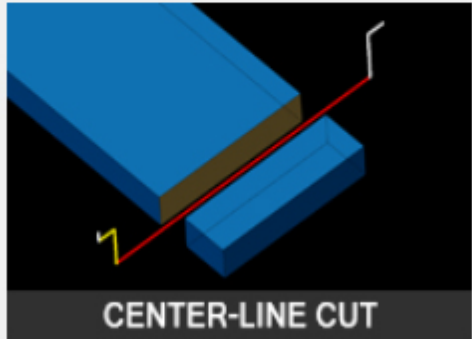
Center-Line Cut will plunge at the beginning of the cut and retract straight up out of the shape at the end of the cut.



Center-Line Cut Tool Path



Center-Line Cut Tool Path

Cycle Information		Status Information	
Offset Dim	0.0	Safety Plane	*0.25000
Cut Side	LH	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Round Corners	N	Feederate/Spindle Speed	
Lead In	N	Feederate	350.00000
Lead Out	N	Spindle Speed	18000.00000
Lead Size	0.0	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
XY Stock Allowance		Calculate	
Z Stock Allowance		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
		 <p>CENTER-LINE CUT</p>	
		Reset Cycle Settings to Default	

Center-Line Cut Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset. In this instance, there should be NO offset so 0 is set by default.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (0.0) is a setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed unless you want to force an offset for the tool path.

See the [Offset Dim](#) section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

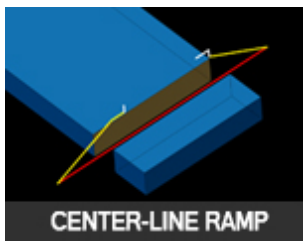
Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

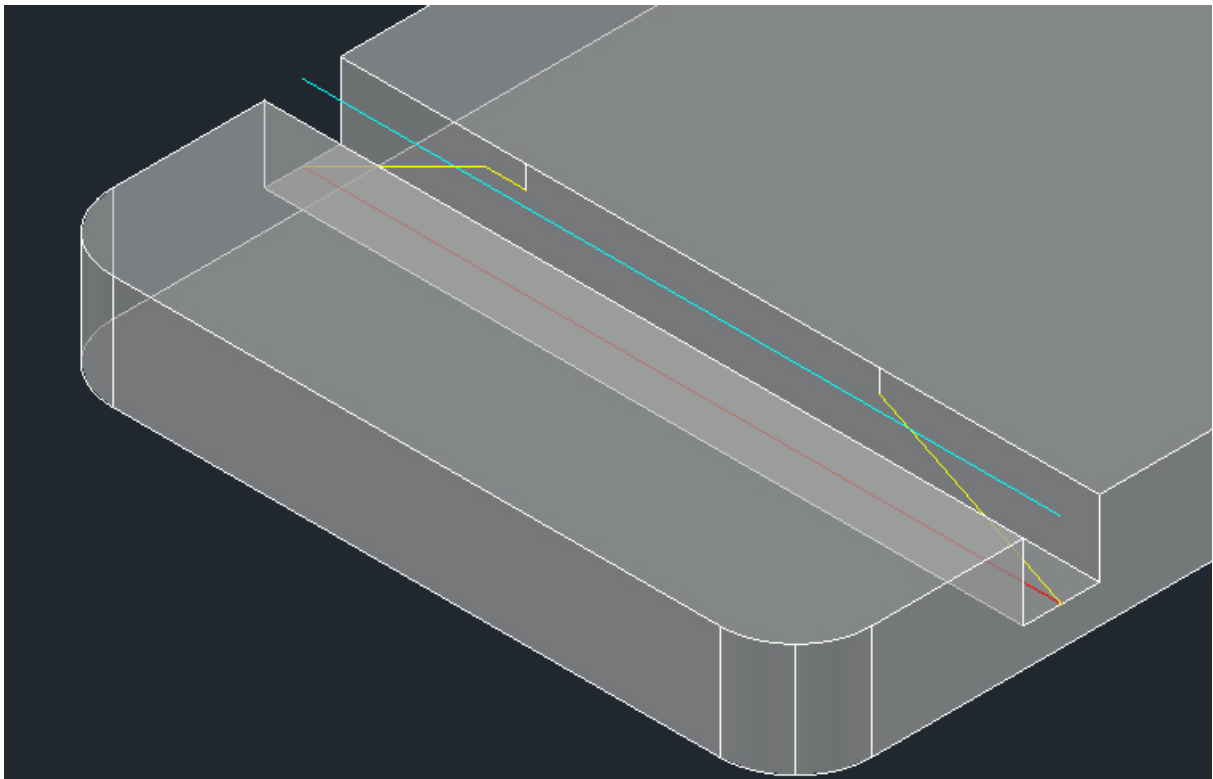
Center-Line Ramp



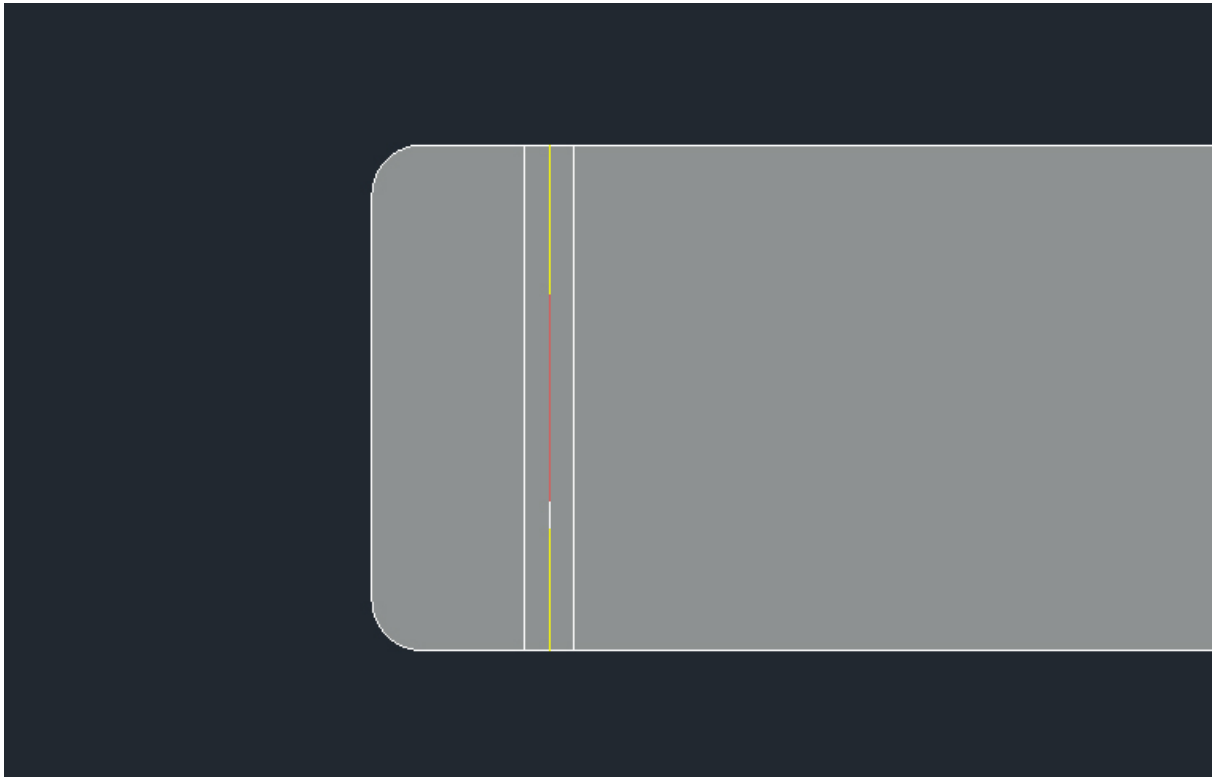
These cycles allow you to create tool paths directly from any defined shapes. Use this cycle to make a fixture or to engrave geometry. This cycle is also used to cut tool-width slots. You can draw the gasket grooves and vacuum grooves for a spoil board, and use this cycle to follow that geometry. This cycle would normally be used with a tool that does not use Machine Cutter Compensation, since there is no offset created.

Cycle Start and End positions determine where to rapid to and where to feed to in the Z axis. Tool Depth of Cut affects the number of cutting passes. You must manually offset the defined geometry to determine the centerline of the tool if you turn on cutter compensation.

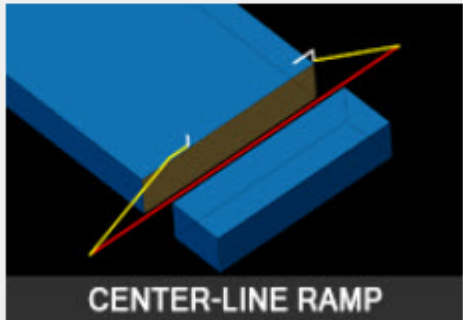
Center-Line Ramp will make a ramp in reverse direction to the cut (so it does not violate the shape) at the beginning and end of the shape. This cycle can be used on open or closed geometry.



Center-Line Ramp Tool Path



Center-Line Ramp Tool Path

Cycle Information		Status Information	
Offset Dim	0.0	Safety Plane	*0.25000
Cut Side	LH	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Round Corners	n	Feedrate/Spindle Speed	
Lead In	N	Feedrate	350.00000
Lead Out	N	Spindle Speed	18000.00000
Lead Size	0.0	Surface FPM	NONE
Lead Ratio	2.0	Units Per Revolution	NONE
Lead Feed	0.5	Ramp Feedrate	
XY Stock Allowance		Calculate	
Z Stock Allowance		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Center-Line Ramp Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset. In this instance, there should be NO offset so 0 is set by default.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (0.0) is a setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed unless you want to force an offset for the tool path.

See the [Offset Dim](#) section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Ratio

Lead Ratio determines the angle of the ramp in Z during the lead in and lead out. You can specify the Lead Ratio as a number that reflects the percentage of the angle from its default. That means that if you want a lead that is twice the normal ramp length (shallower angle) enter 2. If you want a lead that is steeper than the default, enter .5.

See the [Lead Ratio](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

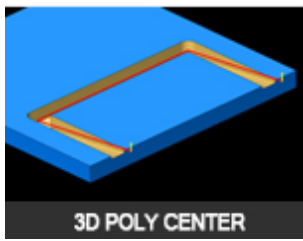
Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings**

3D Poly Center

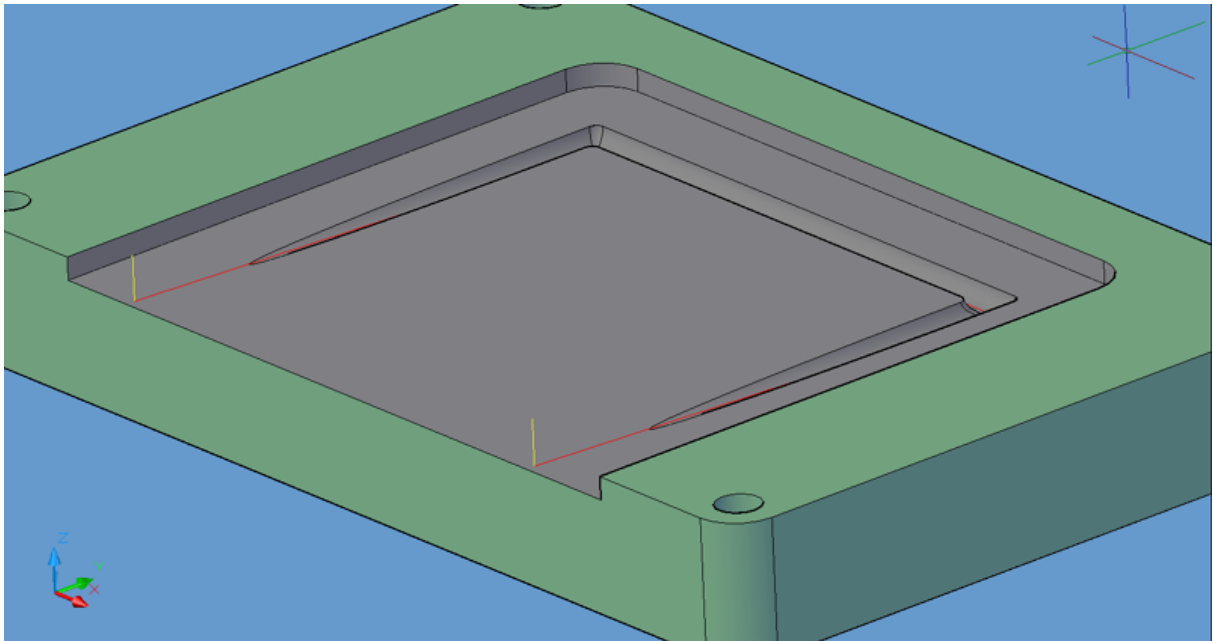


3D Poly Center will follow a 3D Polyline drawn and geoshaped. The tool path will start at the Safety Plane, plunge to the start of the cut and then follow the 3D Polyline to the end and finish by retracting straight up to the Safety Plane.

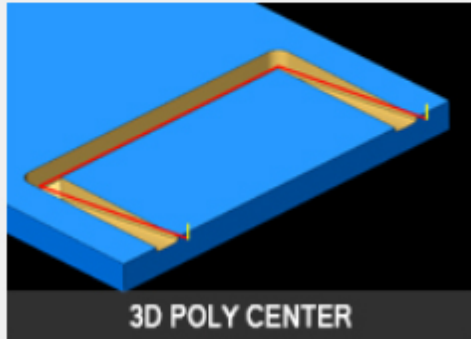
This is a center cutting cycle and no tool radius compensation should be used.

This cycle is used to turn any 3D polyline into a tool path. Draw the 3D polyline on layer NC_SHAPE (or Geoshape a 3D polyline), ensure Cutter Comp is set to No, and click Cut. This cycle will add vertical lead-in and lead-out at the start and end points on the polyline. This cycle is used for cutting sloped slots or varying depth engraving cuts.

If you are drawing in a plane other than WCS in AutoCAD, you will need to Geoshape the geometry if you want to cut from the WCS with this cycle. Router-CIM offers a command that will turn a POLYLINE into a 3D POLYLINE. In your AutoCAD command line, type **2DTO3D** and select on the Geoshape.



3D Poly Center tool path

Cycle Information		Status Information	
Cut Direction	CCW	Safety Plane	*0.25000
Lead Feed		Depth Per Pass	1.00000
XY Stock Allowance		Total Cut Depth	
Z Stock Allowance		Feedrate/Spindle Speed	
		Feedrate	350.00000
		Spindle Speed	18000.00000
		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

3D Poly-Center-Line Parameters

The following parameters effect the toolpath creation:

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

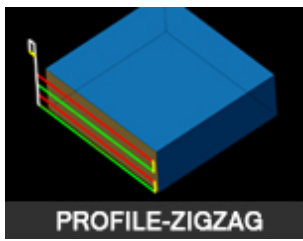
Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

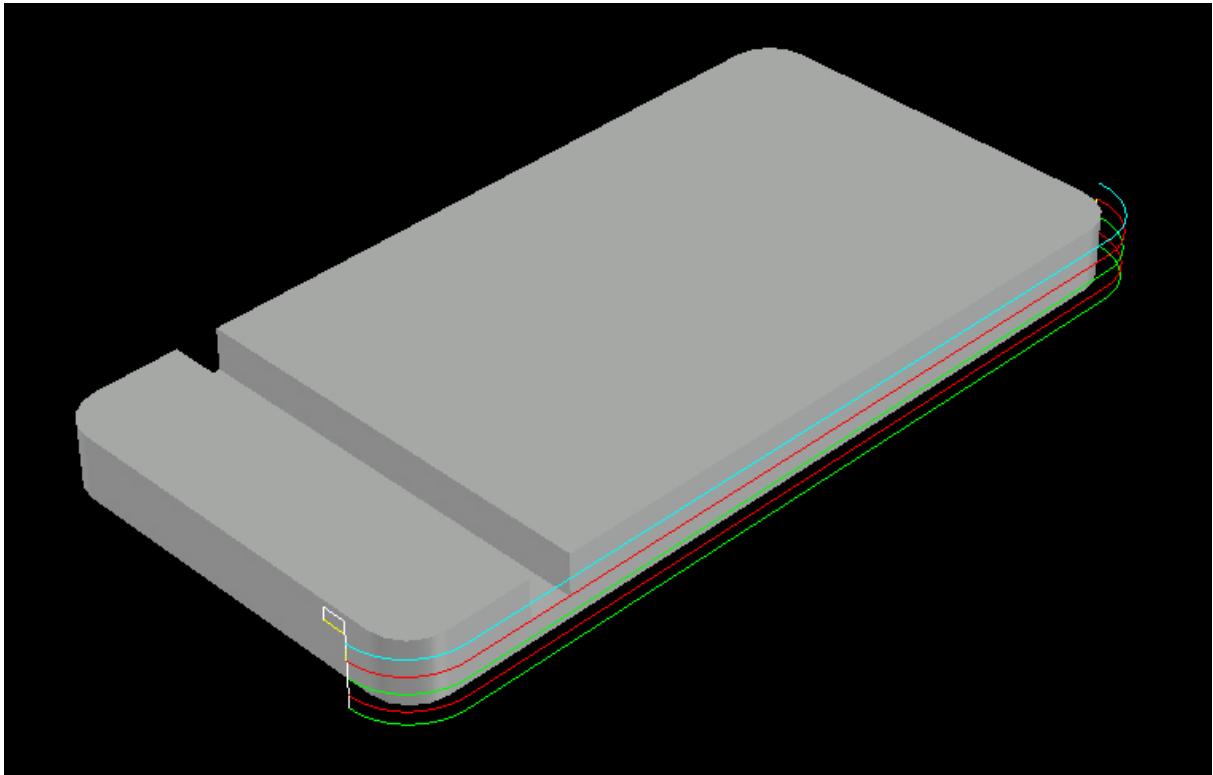
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Profile ZigZag

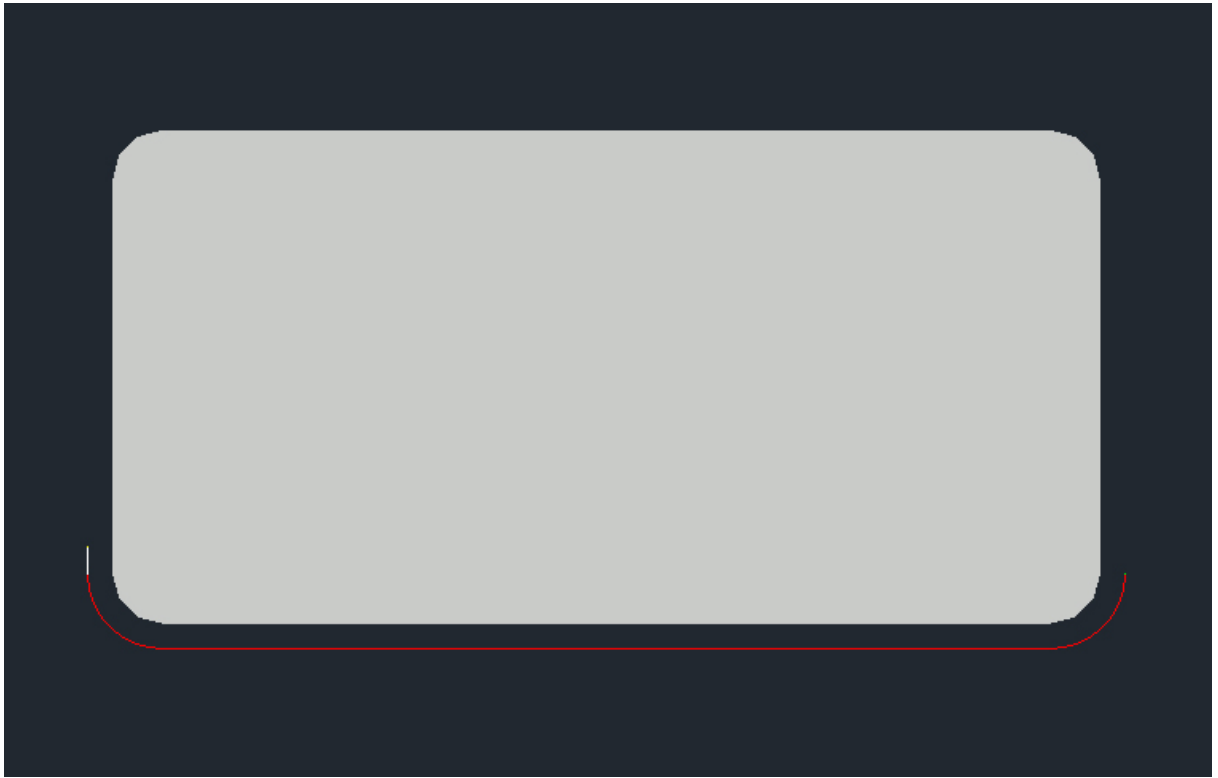


Profile Zig-Zag is a cycle best suited for cutting open shapes. The tool will start at the Safety Plane, plunge to the first cut depth, then follow the shape to the other end, move down in Z to the next depth, and then cut the shape back to the start, repeating the actions until it has reached the Total Cut Depth. Since it is a center cutting cycle, there is no offset by default, so the geometry should be offset or placed in the proper position for the tool. It is possible to have an offset of the tool path, by changing some cycle parameters, described below.


Typically, if a cycle is needed to follow a geoshape and cut multiple depths per pass without the tool lifting at the end of each pass and moving back to the start, then Profile ZigZag is well suited for this cut.



Profile ZigZag Tool Path



Profile ZigZag Tool Path

Cycle Information	Status Information
Lead Feed <input type="text"/>	Safety Plane <input type="text" value="0.25000"/>
	Depth Per Pass <input type="text" value="1.00000"/>
	Total Cut Depth <input type="text"/>
	Feedrate/Spindle Speed
	Feedrate <input type="text" value="350.00000"/>
	Spindle Speed <input type="text" value="18000.00000"/>
	Surface FPM <input type="text" value="NONE"/>
	Units Per Revolution <input type="text" value="NONE"/>
	<input type="button" value="Calculate"/>
	Before Codes <input type="text"/>
	After Codes <input type="text"/>
	Oscillation Amount <input type="text" value="0.00000"/>
	Sort By Rank # <input type="text"/>
	
	<input type="button" value="Reset Cycle Settings to Default"/>

Profile ZigZag Parameters

The following parameters effect the toolpath creation:

Lead Feed

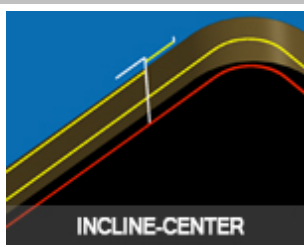
This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the**

toolpath and NC Code carefully before running your machine tool if you change these default settings.

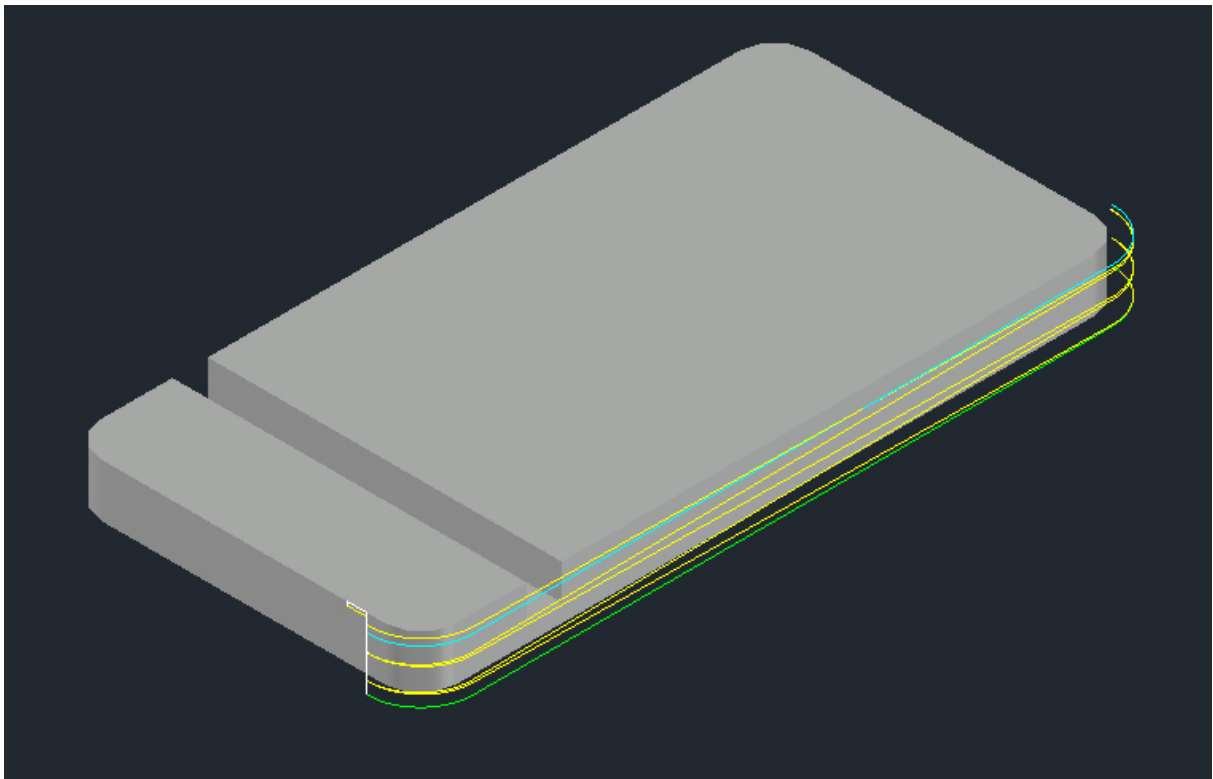
Incline-Center



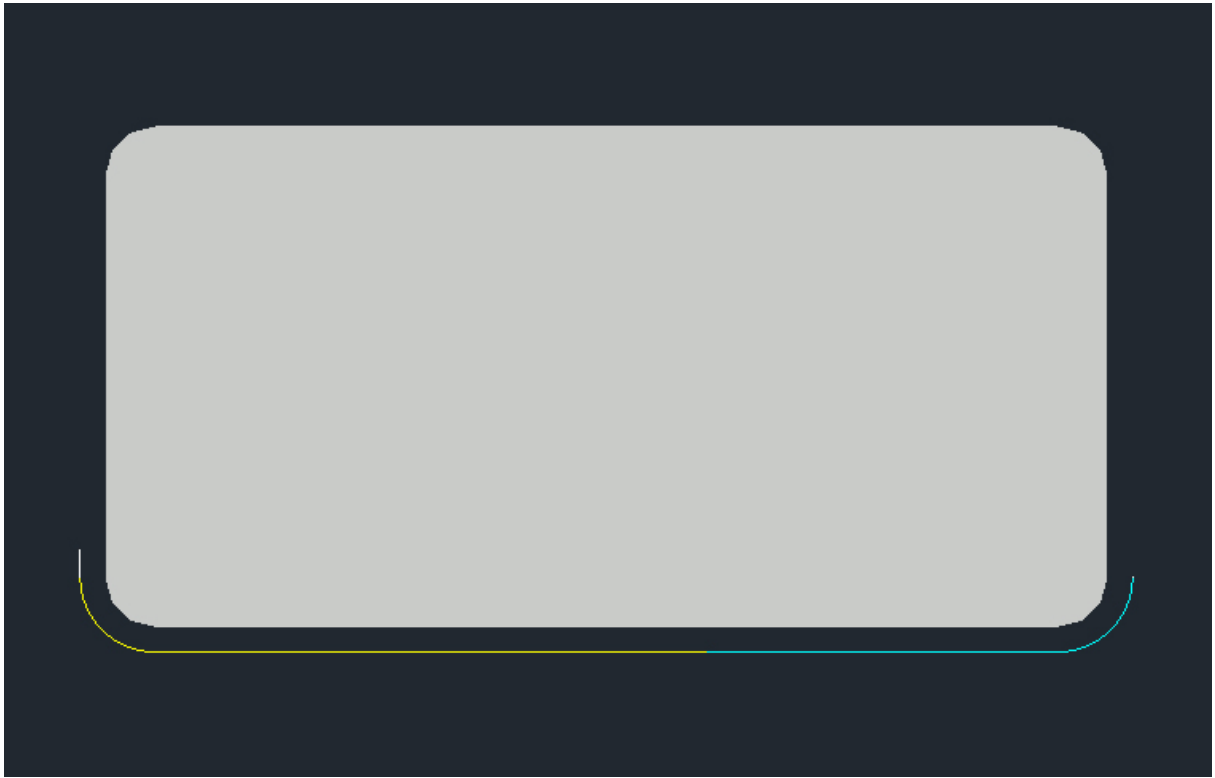
Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.

The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

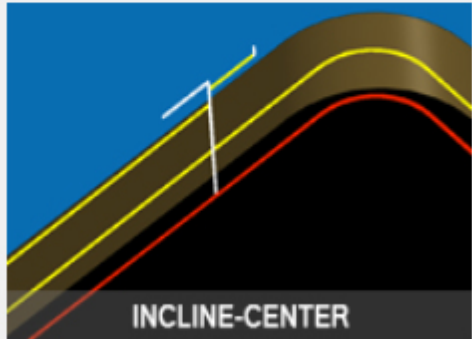
Incline-Center will default to cutting on the centerline of an open or closed shape.



Incline Center Tool Path



Incline Center Tool Path

Cycle Information		Status Information	
Offset Dim	0.0	Safety Plane	*0.25000
Cut Side	LH	Depth Per Pass	1.00000
Cut Direction	INCLCCW	Total Cut Depth	
Round Corners	N	Feedrate/Spindle Speed	
Lead In	N	Feedrate	350.00000
Lead Out	N	Spindle Speed	18000.00000
Lead Size	LEADSCL	Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
XY Stock Allowance		Calculate	
Z Stock Allowance		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Incline-Center Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset. In this instance, there should be NO offset so 0 is set by default.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (0.0) is a setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed unless you want to force an offset for the tool path.

See the [Offset Dim](#) section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

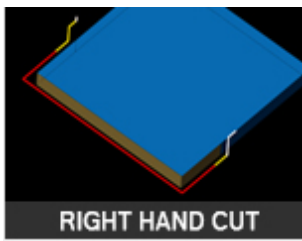
Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Right Hand Cut

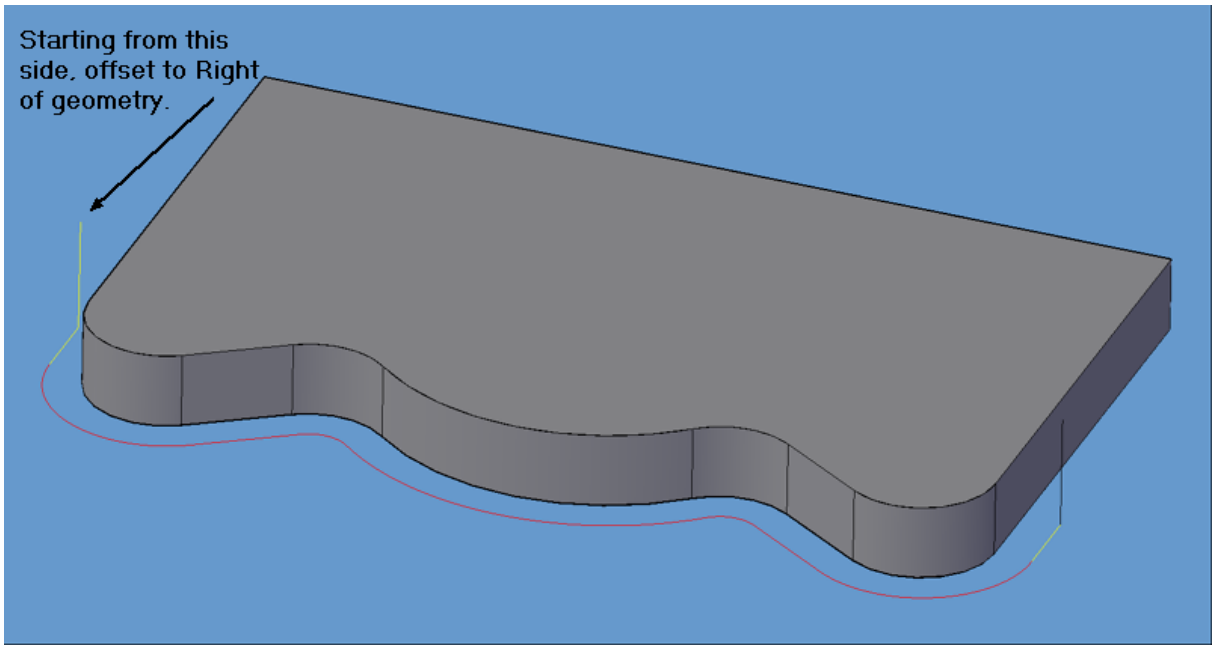


Right Hand Cut is an open shape cut cycle. The cycle will start at the Safety Plane, plunge to the Total Cut Depth and then follow the shape to the end, lead straight out and then retract back to the Safety Plane.

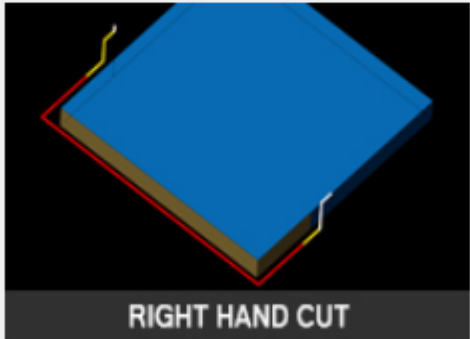
The offset by default is to the right side of the shape, starting from the start point location.

This cycle is only useful in interactive Router-CIM. Automation uses a different method of determining the right from the left side of a shape during geoshape.

Starting from this side, offset to Right of geometry.



Right-Hand Cut Tool Path

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▾	Safety Plane	*0.25000
Cut Side	RH ▾	Depth Per Pass	1.00000
Cut Direction	CCW ▾	Total Cut Depth	
Lead In	"LNTLI" ▾	Feedrate/Spindle Speed	
Lead Out	"LNTLO" ▾	Feedrate	350.00000
Lead Size	LEADSCL ▾	Spindle Speed	18000.00000
Lead Feed	▾	Surface FPM	NONE
XY Stock Allowance	▾	Units Per Revolution	NONE
Z Stock Allowance	▾	<input type="button" value="Calculate"/>	
		Before Codes	▾
		After Codes	▾
		Oscillation Amount	0.00000
		Sort By Rank #	▾
			
		<input type="button" value="Reset Cycle Settings to Default"/>	

Right-Hand Cut Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the [Offset Dim](#) section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

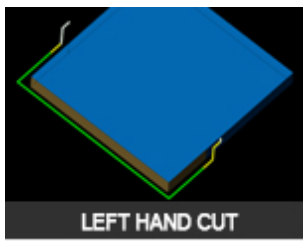
Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

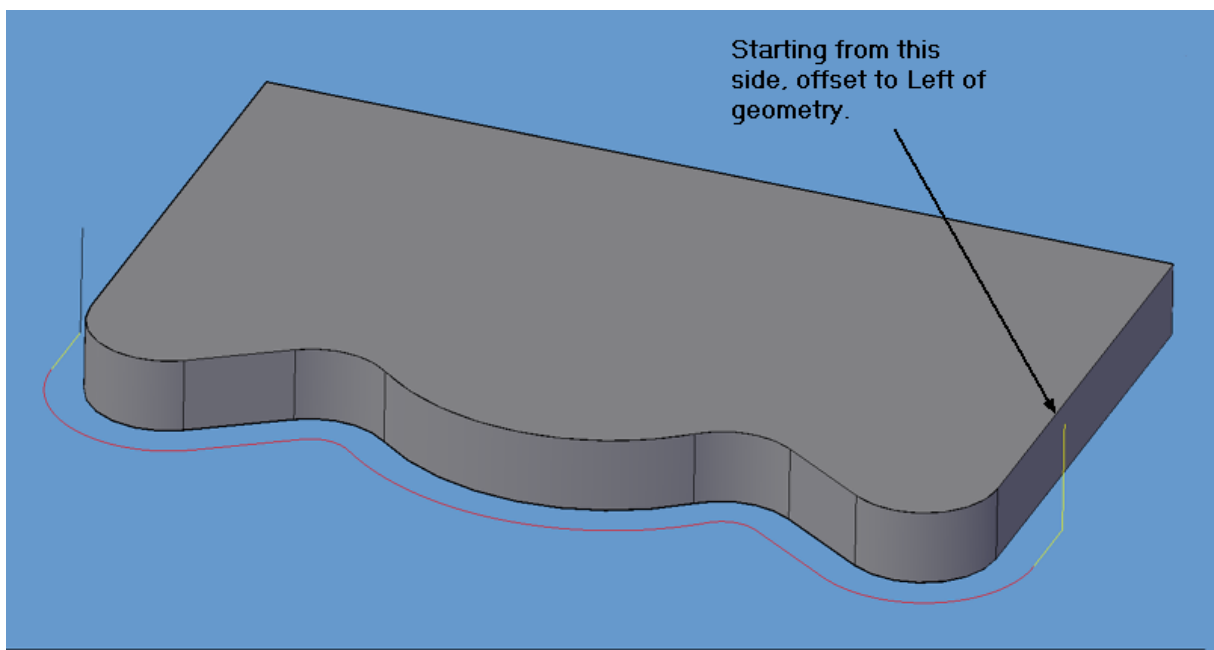
Left Hand Cut



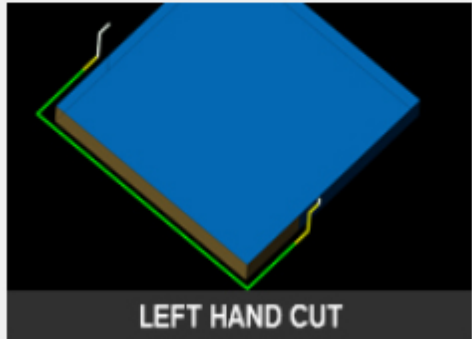
Left Hand Cut is an open shape cut cycle. The cycle will start at the Safety Plane, plunge to the Total Cut Depth and then follow the shape to the end, lead straight out and then retract back to the Safety Plane.

The offset by default is to the left side of the shape, starting from the start point location.

This cycle is only useful in interactive Router-CIM. Automation uses a different method of determining the right from the left side of a shape during geoshape.



Left-Hand Cut tool path.

Cycle Information		Status Information	
Offset Dim	FIRSTXY XYCUTLOC ▾	Safety Plane	*0.25000
Cut Side	LH ▾	Depth Per Pass	1.00000
Cut Direction	CCW ▾	Total Cut Depth	
Lead In	"LNTLI" ▾	Feedrate/Spindle Speed	
Lead Out	"LNTLO" ▾	Feedrate	350.00000
Lead Size	LEADSCL ▾	Spindle Speed	18000.00000
Lead Feed		Surface FPM	NONE
XY Stock Allowance		Units Per Revolution	NONE
Z Stock Allowance		<input type="button" value="Calculate"/>	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		<input type="button" value="Reset Cycle Settings to Default"/>	

Left-Hand Cut Parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the [Offset Dim](#) section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the [Lead Feed](#) section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

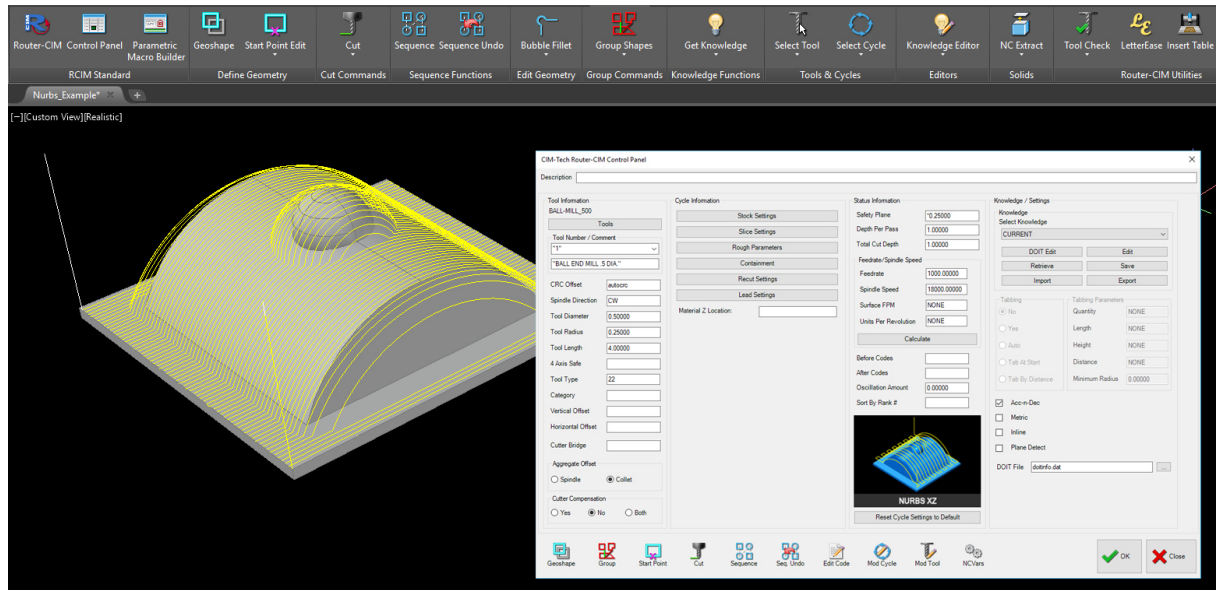
See [Z Stock Allowance](#) for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

NURBS Surface Machining Cycles

The Expert NURBS Cutter Extension was developed to service the complex 3D machining needs of Router users. The 3D Toolpath Calculator is seamlessly integrated with Router-CIM. Toolpaths can be created from 3D solids or surfaces.

Toolpath results from the Expert NURBS Cutter Extension are displayed in the AutoCAD drawing window.



The graphic examples used in this section of the manual were captured from AutoCAD.

Overview:

Router-CIM communicates with the toolpath calculator, or the Expert NURBS Cutter, by using files external to the drawing. You may find some, or all, of the following files being created when using the Expert NURBS Cutter.

"Filename.GEO" contains the NURBS description.

"Filename.PAR" contains parameter controls.

"Filename.O" contains toolpath data that is read by NC Polaris when toolpath is being returned to the drawing.

When multiple machining steps are required for the same group of surfaces or solid (i.e., roughing and finishing), you would run "Cut" multiple times. After running "Cut" the first time, the system recognizes that the ".GEO" file already exists for the drawing name and during subsequent "Cut" requests you are prompted, "Define Surfaces YES/NO". If the operation is intended to address the identical surfaces (or solid), using identical surface tolerances and complimenting entities, you can respond "NO", to the prompt. If a different combination of surfaces (or solid), surface(s) tolerances, or different complimenting entities are required, you must respond "YES", and select the new combination of surfaces (or solid) and complimenting entities for the next operation. Defining Surfaces is a relatively fast function, and it is always safer to respond "YES", and select the surfaces again.

As a general housekeeping rule, after completing an entire job, you can delete the external files to avoid unnecessary buildup of files on the hard disk drive in your computer.

Surface Construction Rules:

A surface must not contain C0 continuity. If unusual toolpath results occur on a given surface, use AMD's Surface Editing Commands to break at C0 to ensure proper performance from the Expert NURBS Cutter Extension.

Occasionally IGES translators will create surface or solid anomalies. Be prepared to use AMD editing features to repair these surface or solid anomalies when unusual toolpath results.

Surfaces created from the 3D Solid should be truncated to ensure proper operation of the Expert NURBS Cutter Extension. Toolpath can be produced directly from the 3D Solid.

A Cavity (female part) must contain a horizontal plane(s) that represents the top of the Cavity or Parting Plane. This Plane(s) can be very narrow to avoid unnecessary toolpath motions. Without the Parting Plane(s) the software can not establish an approach to vertical surfaces, or surface areas that approach near vertical at termination in a Cavity, and results can be unpredictable. A narrow horizontal plane at the top of Cavity wall gives the software an established location to start the toolpath.

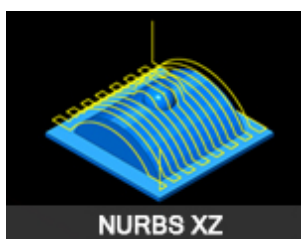
NURBS Cycles Supplied

The Expert NURBS Cutter Extension can be operated by using one of the Cycles in the Cycle List containing the word "XNURBS". The appropriate Cycle, used in conjunction with entries in the Knowledge Editor GUI, will generally provide an intuitive method for entering machining parameters. All five XNURBS Cycles share the same parameters. The differing default parameter entries cause them to perform differently.

The five Cycles supplied are:

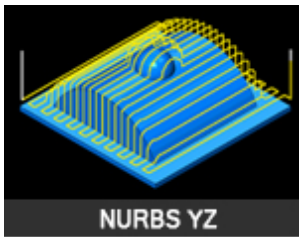
NURBS CUT-ZIGZAG XZ

Default settings will produce vertical toolpath at a constant step over equal to tool radius in the XZ plane (angle 0).



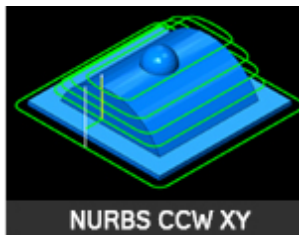
NURBS CUT-ZIGZAG YZ

Default settings will produce vertical toolpath at a constant step over equal to tool radius in the YZ plane (angle 90).



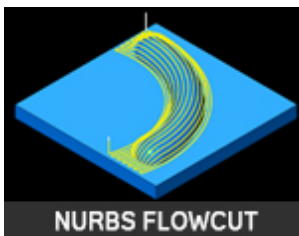
NURBS CUT-CCW XY

Default settings will produce horizontal climb milling toolpath at a constant step over equal to tool radius in the XY plane.



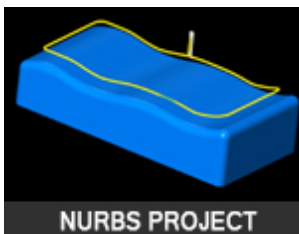
NURBS CUT-FLOWCUT

Default settings will produce vertical flowcut toolpath at a constant step over equal to tool radius in the XZ plane (angle 0) or YZ plane (angle 90).



NURBS CUT-PROJECT

Default settings will produce vertical toolpath that follows the direction of the projection polyline.



This section of the manual is supported by a series of graphics, showing tool path results in the drawing. Each of the NURBS Cycles are used. A variety of entries in the Knowledge Editor will give different toolpath results. Drawings MCADTPC.DWG, and MCADFLOW.DWG, will be used to describe the capabilities of the NURBS Cutter. The toolpath results shown in all examples were created by using a.25 Diameter Ball End Mill.

The NURBS Cutter supports machining with Ball, Bull or Flat End Mill Tools, on solids or surfaces. Once you have familiarized yourself with the operation of the NURBS Cutter using the .25 diameter Ball End Mill Tool, you are free to experiment using Bull or Flat End Mill Tools.

Drawings used for the examples will be made available via hyperlinks so that you may download the completed drawings if desired.

NURBS Cycle Parameters

The Expert NURBS Cutter is rich in 3D machining features. The Mill Cycle List has been expanded to include a special parametric table to accommodate the Expert NURBS Cutter. The Expert NURBS Cutter parametric table is described as follows. Default parameter entries are shown for the MILL_XNURBS_CUT-ZIGZAG_XZ Cycle. Graphic toolpath examples using each of the XNURBS Cycles with different Knowledge Editor entries are shown in the EXAMPLES section.

Your focus when operating the Expert NURBS Cutter will be entering values in the Knowledge Editor. The following list of Cycle Parameters is for reference when altering parameter defaults, or making additional Cycles.

[01] Enter Table Name..... mltpc

Table name. Can not be edited.

[02] Enter Item Name..... XNURBS_CUT-ZIGZAG_XZ

Cycle name. Editing name enables making of new Cycles.

[03] Cycle Type.....(MLTPC) MLTPC SCUT2

Defines the type of cutting cycle and should not be edited.

[04]

Reserved, do not edit.

[05] Vert/Horz/Flow/Proj Cut V

Enter V for Vertical cutting, H for Horizontal cutting, F for Flow cutting, and P for polyline projection cutting.

[06] Rough Z Cut Increment (#)

An entry in this parameter automatically enables roughing. The value entered controls the maximum cutting depth for each roughing pass.

[07] Roughing Type (1/2/3)

A total of 6 roughing methods are available with the Expert NURBS Cutter. They are divided into two categories, Vertical and Horizontal.

Vertical Cutting + Mode 1 = Offsets each roughing pass by [06] "Rough Z Cut Increment" + [22] "Stock Amount to Leave", moving in rapid when machine motions exceed [19] "Working Area Z Maximum".

Vertical Cutting + Mode 2 = Offsets each roughing pass by [06] "Rough Z Cut Increment" + [22] "Stock Amount to Leave", and proportionally distributes Z depth throughout the roughing activity without allowing tool to leave the stock material.

Vertical Cutting + Mode 3 = Shifts roughing motions in "Z" based on [06] "Rough Z Cut Increment" while compensating for [22] "Stock Amount to Leave", moving in rapid when machine motions exceed [19] "Working Area Z Maximum".

Horizontal Cutting + Mode 1 = Plunges straight down to each Z level, compensates for [22] "Stock Amount to Leave", roughs without following collective surface contours before proceeding to next lower Z level.

Horizontal Cutting + Mode 2 = Ramps down to each Z level, compensates for [22] "Stock Amount to Leave", roughs and makes a contour following pass before ramping down to next Z level.

Horizontal Cutting + Mode 3 = Plunges straight down to each Z level, compensates for [22] "Stock Amount to Leave", roughs and makes a contour following pass before plunging straight down to next Z level.

[08] Surface Intersect (Y/N)

Y enables intersect cutting. Intersect cutting can be applied to all surface intersections in a drawing, or applied only to the intersections of selected sets of surfaces. If the surface drawing has already been defined to the system by previously making toolpath (a.GEO already exists), respond YES to the "Define Surfaces" prompt. You are then prompted "Group Surfaces into Sets?".

1. If your objective is to apply surface intersect toolpath on all surface intersections in the drawing, you should respond NO, and select all surfaces in the drawing when prompted to select objects.
2. If your objective is to apply surface intersect toolpath on intersections of selective sets of surfaces, you should respond YES. You are then given the opportunity to "Select Surfaces in First Set" and "Select Surfaces in Second Set". The "First Set" will disappear from the screen when you have completed the selection process for user feedback.
3. Upon completion of selecting objects you are prompted to, "Specify Minimum Angle that Defines Intersection Deg:". Respond according to your requirements.

[09] CrossCut.....(Y/N)

Y enables CrossCut. Cross cutting only applies to vertical finishing motions. Cutting motions are produced on any surface(s) area(s) that meet the angle criteria, in the direction of the cutting motions. You are prompted during "Cut" for surface(s) angle criteria.

[10] Recut.....(Y/N)

Y enables Recut. Recut, sometimes referred to as rest cutting, removes material left by a prior operation or tool. In Vertical cutting mode (entry V in [05]), the system will prompt for information regarding previous tool used. Toolpath is then produced on all surface areas not cut by the prior tool. In Horizontal cutting mode (entry H in [05]), no prompt occurs regarding previous tool, and vertical toolpath is produced on all surface areas that are nearly horizontal where XY motions could not be generated.

[11] Define Contain Surf(Y/N)

Y enables Surface Containment. If the surface drawing has already been defined to the system by previously making toolpath (a .GEO file exists), respond YES, to the "Define Surfaces" prompt. You are prompted during "Cut", to select surfaces to cut, followed by selection of surfaces to use for toolpath containment. This feature is not available when cutting directly from a solid.

[12] Def. Contain String(Y/N)

Y enables String Containment. If the surface drawing has already been defined to the system by previously making toolpath (a .GEO file exists), respond YES, to the "Define Surfaces" prompt. You are prompted during "Cut" to select surfaces to cut and to select 2-D polyline(s) to use to contain toolpath. The XY boundary of a single closed polyline will contain the toolpath limits.

[13] Define Plunge Pts.(Y/N)

Y enables Plunge points. Plunge points apply only to roughing. In the roughing mode the system will make every attempt to plunge at the defined plunge locations. You are prompted during "Cut" to define plunge points.

[14] Working Area X Minimum.

[15] Working Area Y Minimum.

[16] Working Area Z Minimum.

[17] Working Area X Maximum.

[18] Working Area Y Maximum.

[19] Working Area Z Maximum.

Parameters [14] through [19] are used to define a working area (clip box). Normal system operation requires that parameter [19] "Working Area Z Maximum", contains a value equal to the value entered in parameter [21] "Material Z Loc(#)". Entry of a value different than the entry in [21] "Material Z Loc.#", is used only under special conditions.

[20] Description.....("TEXT") "X-Nurbs Cutting, Zigzag XZ Plane"

Text string used to describe Cycle.

[21] Material Z Plane Loc.(#)

Requires entry of where the top of the stock material is located. Cycle start position uses this entry to calculate feed distance to material. Normal system operation requires that parameter [19] "Working Area Z Maximum", contains a value equal to the value entered in parameter [21] "Material Z Loc(#)".

[22] Stock Amount to Leave... 0.0 Y

The first entry controls the amount of stock to leave. The second entry "Y" (without the quotes), indicates that you would like multiple stock allowances. You are prompted during CUT to select surface(s) for different stock allowance, and you are then prompted for a value.

[23] Slice Step Dir (POS/NEG) POS

Determines whether vertical cutting motions begin at X or Y zero and proceed positive, or begin at X or Y maximum and proceed negative.

IMPORTANT NOTE<R>The determination for CONVENTIONAL or CLIMB milling is automatically made by designating a POS or NEG answer to this parameter when using Horizontal finishing, as long as there is no entry in Parameter [29]CW Horz. Cuts....(Y/N). A POS answer results in CONVENTIONAL milling, a NEG answer results in CLIMB milling. An entry of Y or N in Parameter [29]CW Horz. Cuts....(Y/N), will override the automatic computation.

[24] Constant Step Over Dist. !*tr*

This entry defines the fixed distance between cuts. This entry is overridden when [25] "Define Scallop Size" is enabled with a Y entry.

[25] Define Scallop Size(Y/N)

Y enables Scallop Height Control for Vertical finishing. You are prompted during "Cut" for desired scallop height. Enter the desired scallop height at that time. You are then prompted for minimum step size. Minimum step size is a clamp to protect from excessive toolpath on unusually steep surface areas. A response of .001 for Inch, or .01 for Metric, is appropriate in most cases. A response too large will circumvent scallop height calculation.

[26] Vertical Cutting Angle. 0.0

Defines angle of Vertical toolpath. 0.0 = XZ, 90 = YZ. Any angle between 0 and 90 is valid. Negative angles are allowed. This parameter entry works in conjunction with [23] "Slice Step Dir (POS/NEG).

[27] Lace Cutting Paths (Y/N) Y

Y enables lace or zigzag cutting motions in Vertical and Horizontal cutting. Enter N to achieve uni-directional cuts in Vertical cutting. Enter N to achieve continuous direction cutting when using Horizontal cutting.

[28] Fall Over Mode (0/1/2/3)

Fallover Mode settings accommodate special Vertical finishing situations. The system default is Fallover Mode 3. In Fallover Mode 3, vertical finishing toolpath ends at the extents of the selected surfaces. Fallover Mode 1 and 2 cause toolpath to roll over the extents of the selected surfaces by tool radius. Mode 2 stops at tool radius, Mode 1 continues down in Z to "Working Area Z Minimum". Fallover Mode 0 causes toolpath to roll over as in mode 1, but also move away from the surface by a distance equal to tool radius. Fallover Mode settings are managed by the system for normal operation.

[29] CW Horz. Cuts.....(Y/N)

This parameter applies only to Horizontal finishing. The default is no entry. No entry allows the entry in [23]Slice Step direction to automatically control CLIMB or CONVENTIONAL cutting. N forces counter clock wise cutting. Y forces clock wise cutting direction. CCW toolpath is RED and CW toolpath is GREEN.

Lace cutting set to Y will cause zigzag (back and forth) horizontal finishing motions. This is particularly useful when cutting single or open multiple surfaces. Lace cutting should be set to N for normal multiple surface finishing.

[30] Lead-In Name ("NAME"/N)

[31] Lead-Out Name ("NAME"/N)

[32] Size of Leads.....(#)

[33] Angle of Leads.....(#/N)

[34] Task @ Lead-In (name/N) TPC-FEED

[35] Task @ Cut Start (name/N) TPC-CUT

[36] Task @ Cut End (name/N)

[37] Task @ Lead-Out (name/N)

[38] Run Tasks in Slices(Y/N) Y

Parameters [30] through [38] follow normal Router-CIM convention.

[39]

[40] (reserved).....

Reserved

[41] Task @ Rough Str(name/N)

[42] Task @ Rough End(name/N)

Parameters [41] and [42] follow normal Router-CIM convention

[43] Complete NC Program(Y/N) Y 1234

Y enables Complete NC Program which causes Router-CIM to go directly to NC Code after producing toolpath. The numeric value entered represents the Job Id.

[44] STR/END Tasks for NC Prg

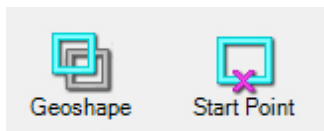
This parameter allows the use of Start/End Codes when "[43]Complete NC Program(Y/N)" is enabled. Entry in this parameter to enable Start/End Codes is: PROCTSK1 PROCTSK2

Knowledge Editor For NURBS

The screenshot shows the 'CIM-Tech Router-CIM Control Panel' window. The 'Description' field is empty. The 'Tool Information' section on the left includes fields for 'Tools' (BALL-MILL_500), 'Tool Number / Comment' (1), 'CRC Offset' (autocr), 'Spindle Direction' (CW), 'Tool Diameter' (0.5000), 'Tool Radius' (0.2500), 'Tool Length' (4.0000), '4 Axis Safe', 'Tool Type' (22), 'Category', 'Vertical Offset', 'Horizontal Offset', 'Cutter Bridge', 'Aggregate Offset' (Spindle/Collet), and 'Cutter Compensation' (Yes/No/Both). The 'Cycle Information' section in the middle includes buttons for 'Stock Settings', 'Slice Settings', 'Rough Parameters', 'Containment', 'Recut Settings', and 'Lead Settings', along with a 'Material Z Location' field. The 'Status Information' section on the right includes fields for 'Safety Plane' (0.25000), 'Depth Per Pass' (1.0000), 'Total Cut Depth' (1.0000), 'Feedrate/Spindle Speed' (Feedrate: 1000.0000, Spindle Speed: 18000.0000), 'Surface FPM' (NONE), 'Units Per Revolution' (NONE), and a 'Calculate' button. Below this is a 3D model of a NURBS surface labeled 'NURBS XZ'. The 'Knowledge / Settings' section on the far right includes a 'Knowledge' dropdown (CURRENT), buttons for 'DOIT Edit', 'Edit', 'Retrieve', 'Save', 'Import', and 'Export', 'Tabbing' options (No, Yes, Auto, Tab At Start, Tab By Distance), 'Tabbing Parameters' (Quantity, Length, Height, Distance, Minimum Radius), checkboxes for 'Acc-n-Dec', 'Metric', 'Inline', and 'Plane Detect', and a 'DOIT File' field (dotinfo.dat). At the bottom, there is a toolbar with icons for Geoshape, Group, Start Point, Cut, Sequence, Seq. Undo, Edit Code, Mod Cycle, Mod Tool, and NCVars, along with 'OK' and 'Close' buttons.

As you can see, the Control Panel changes into the Knowledge Editor when a NURBS cycle is selected. The NURBS configuration of the Knowledge Editor is customized to support the operation of the NURBS Cutter Cycle Editing. An explanation of the NURBS configuration of the Knowledge Editor follows.

Geoshape and Edit Start Point can be used to control polyline direction and start point alignment when using Flow Cutting. Flow Cut Polylines must be going in the same direction. Closed Flow Cutting requires Start Point alignment. The Router-CIM command NCREVDIR will reverse the direction of Polyline. Flow line toolpath follows the direction of the Flow Polyline.



The Safety Plane entry defines the safe Z clearance plane.

Note: Depth Per Pass and Total Cut Depth are used as input for NURBS Project Cutting to limit cut depth per pass and total cut depth only and is not active with all other NURBS cut cycles.

Safety Plane	<input type="text" value="0.25000"/>
Depth Per Pass	<input type="text" value="1.00000"/>
Total Cut Depth	<input type="text" value="1.00000"/>

The Material Z Location entry indicates the location of the top of the stock material.

Material Z Location:	<input type="text"/>
----------------------	----------------------

The Stock Settings Button opens an Edit Window for entry of Stock Allowance, sometimes referred to as finish allowance. A Check Box ON indicates that certain surfaces will require a different stock allowance(s). Multiple Stock Allowances are allowed.

Stock Settings

×

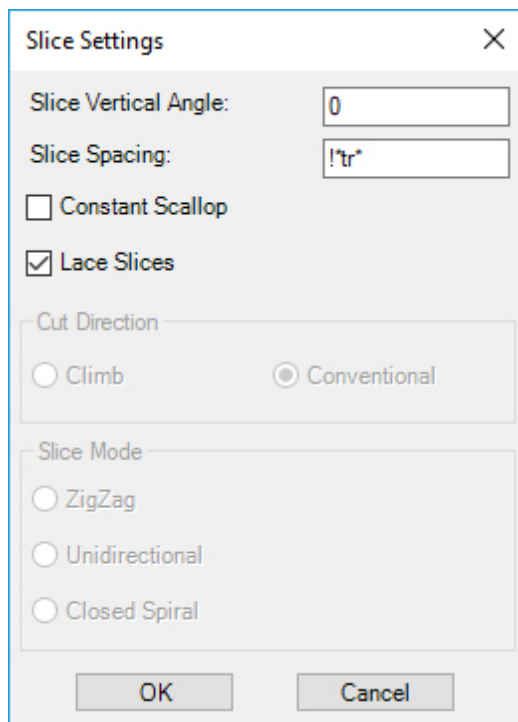
Stock Allowance:

☐ Vary Stock Allowance

OK

Cancel

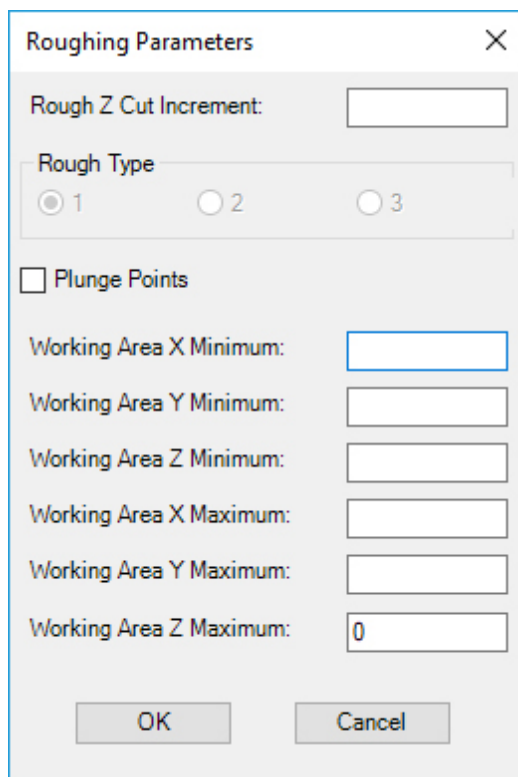
The Slice Settings Button opens an Edit Window for entry of Vertical Cutting Angle, Slice Spacing, Constant Scallop Toggle, and Lace or Zig-Zag Slices. If Constant Scallop is selected you are prompted for maximum scallop height during CUT. If Lace Slices is toggled on, toolpath will Zig-Zag. If toggled off, toolpath will be Unidirectional.



The **Slice Settings** dialog box contains the following controls:

- Slice Vertical Angle:** A text input field with the value `0`.
- Slice Spacing:** A text input field with the value `!tr*`.
- Constant Scallop:** An unchecked checkbox.
- Lace Slices:** A checked checkbox.
- Cut Direction:** A group box containing two radio buttons: **Climb** (unchecked) and **Conventional** (checked).
- Slice Mode:** A group box containing three radio buttons: **ZigZag** (unchecked), **Unidirectional** (unchecked), and **Closed Spiral** (unchecked).
- Buttons:** **OK** and **Cancel** buttons at the bottom.

The Rough Parameters Button opens an Edit Window for entry of roughing parameters. Rough Z Cut Increment controls the maximum depth of each roughing pass. Rough Type controls the type of roughing motions desired.



The **Roughing Parameters** dialog box contains the following controls:

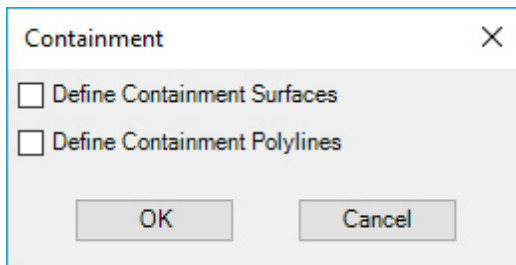
- Rough Z Cut Increment:** A text input field.
- Rough Type:** A group box containing three radio buttons: **1** (checked), **2** (unchecked), and **3** (unchecked).
- Plunge Points:** An unchecked checkbox.
- Working Area X Minimum:** A text input field.
- Working Area Y Minimum:** A text input field.
- Working Area Z Minimum:** A text input field.
- Working Area X Maximum:** A text input field.
- Working Area Y Maximum:** A text input field.
- Working Area Z Maximum:** A text input field with the value `0`.
- Buttons:** **OK** and **Cancel** buttons at the bottom.

A total of 6 roughing methods are available with the Expert NURBS Cutter. They are divided into two categories, Vertical and Horizontal. When the XZ or YZ Cycle is selected, you are in Vertical Cutting. When the XY Cycle is selected you are in Horizontal Cutting.

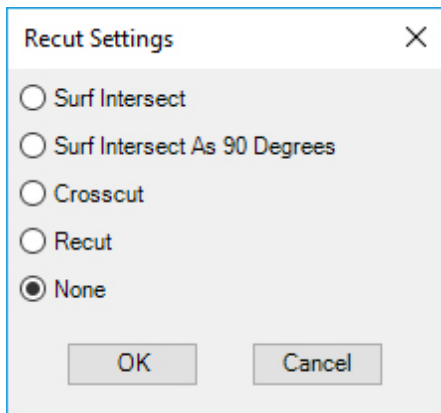
Note: Roughing Parameters are not available for NURBS Flow and NURBS Project cuts.

Plunge Points allows user selection of plunge locations. Always provide multiple plunge point locations. The system may create alternate plunge locations. Plunge points can be located inside or outside of the part.

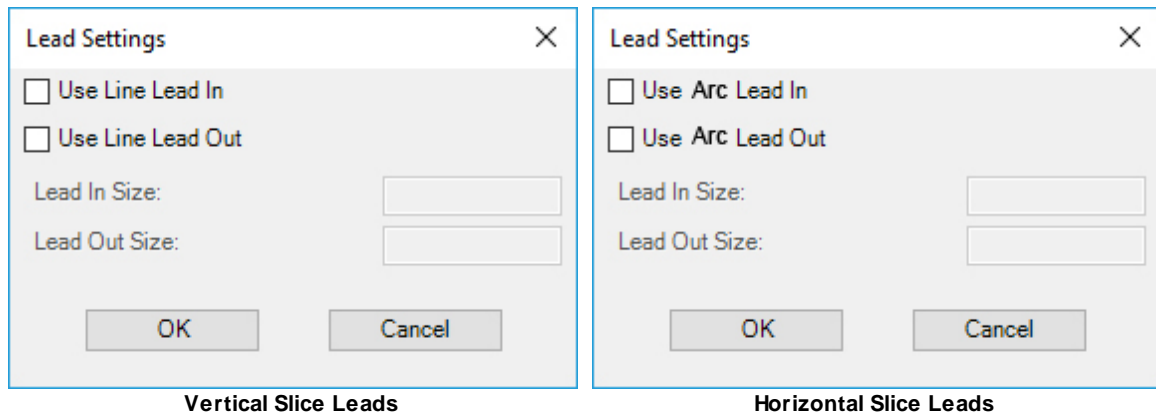
The Containment Button opens an Edit Window for selection of Surface Containment or Polyline Containment.



The Recut Settings Button opens an Edit Window for selection of Recut methods. Surface Intersect produces toolpath at surface(s) intersection(s). Crosscut produces toolpath on surfaces that meet angle criteria. Recut is used to produce toolpath, using a tool smaller than the previous tool, in areas that the larger tool could not access.



The Lead Settings Button is to generate a line lead in and a line lead out of the tool path. The Button opens an Edit Window for selection of Line Lead-In/Out's when using XZ or YZ Cutting. Lead size is controlled by the Lead Size entry. Leads do not apply to XY Roughing. The Button opens an Edit window for selection of Arc Lead-In/Out's when using XY Cutting.



Tools in XNURBS

The NURBS Cutter supports BALL, BULL and FLAT End Mill Tools. Certain tool parameter entries are critical in order to achieve desired results.

Tool Type

The following tool parameter entries are used to calculate proper toolpath for the three types of tools supported.

Parameter [16]Tool Type (21,22,13,23)

Flat End Mill = 21

Ball End Mill = 22

Bull End Mill = 23

Drill = 13

Parameter [19]Tool Tip Dist to Zero(#)

This parameter will loft the tool path by the amount shown.

Flat End Mill = enter 0.0.

Ball End Mill = enter value of ball radius.

Bull End Mill = enter value of bull nose corner radius.

Parameter [19], "Tool Tip Dist to Zero(#)"

is also used to define corner radius of the tool for Bull End Mills.

Parameter [21]Tool Diameter *TW* (#)

Enter tool diameter regardless of tool type.

Parameter [23]Tool Radius *TR* (#)

Enter tool radius regardless of tool type.

Parameter [24]Tool Cut Depth *TD* (#)

Enter maximum cut depth per machining pass regardless of tool type.

Establishing The Feed Plane

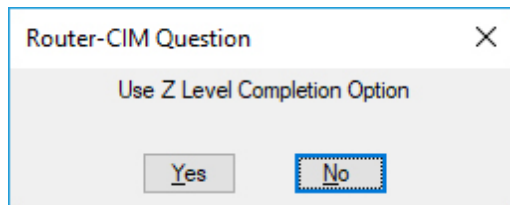
The default Feed Plane of the Expert NURBS Cutter is based on the entry in Material Z Location of the Knowledge Editor. Feed Distance to Material (Tool Editor entry) is added to the entry in Material Z Location. An option exists to allow the Feed Plane to vary, based on where the next X or Y cutting motion(s) begins. The Feed Distance to Material entry is added to this position instead of the Material Z Location. This option is controlled by the NC Variable "NCS_CutPlaneMode". When "NCS_CutPlaneMode" is set to True (T) the toolpath will output rapid motions to within the value of Feed Distance to Material from where material is calculated to be already removed.

To change this NC Variable type NCVAR, select SYSTEM, and change "NCS_CutPlaneMode" to T. Although cycle time can be longer when feeding begins at Material Z Location it is generally considered safer to avoid rapid motions below this plane. You choose the method that best suits your company's policies and procedures. You can change this NC Variable at any time. The Examples in this tutorial are all run with this NC Variable set to nil.

Z Level Completion Option

Z Level Completion is used when you wish the tool path to move up to the clearance plane between each pass. Effectively completing one pass in Z, then retracting, moving to the start of the next pass, plunging and then finishing that Z level cut and retracting to the safety plane, and repeating the process.

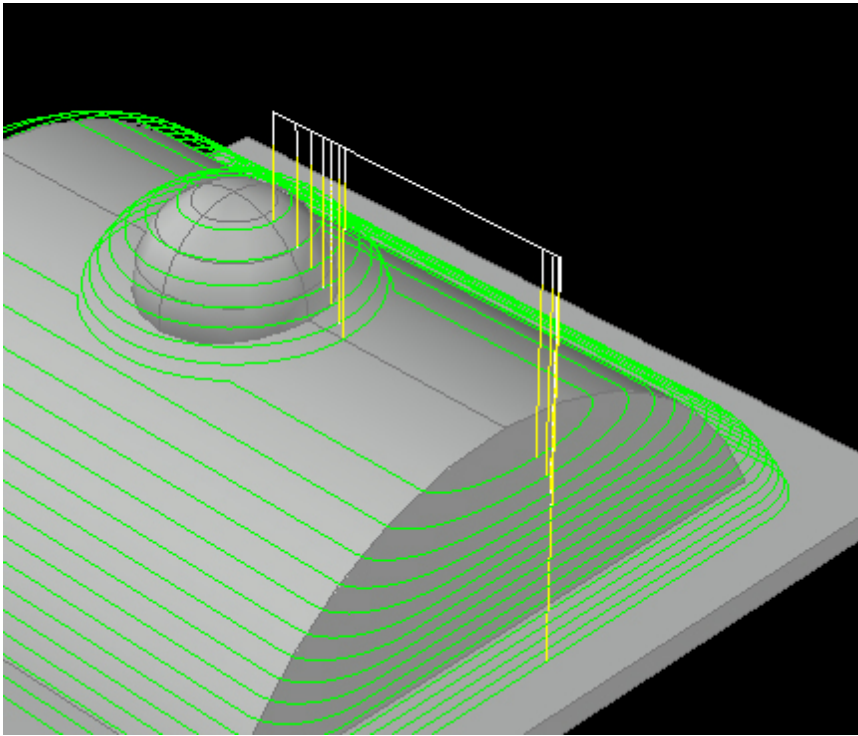
The Prompt will appear on the screen when using the NURBS_CCW_XY horizontal slicing cycle.



YES is used when a Cavity or Core contains multiple hills and valleys. YES will cause the toolpath calculator to complete all toolpath(s) required at each Z level, retract, and then go down to the next Z level.

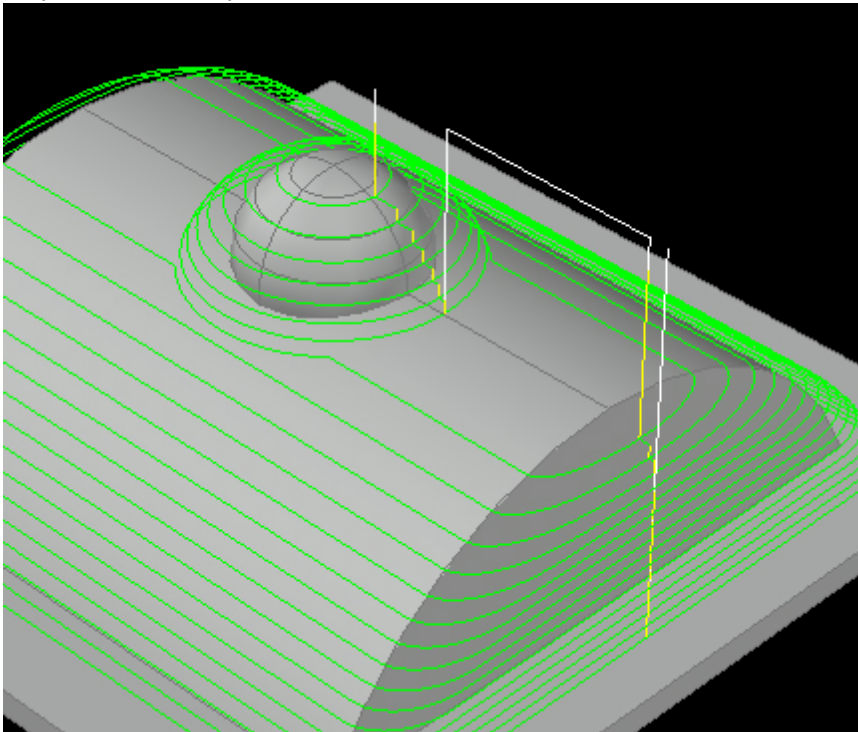
NO continues to cut downward until complete.

Yes produces a tool path similar to this:



Where the tool is retracting between each slice it takes in XY at the current Z plane.

No produces a tool path similar to this:



Once the tool completes a slice in the XY at the current Z plane, it simply moves down to the next Z plane and starts another slice in XY.

Fall over Mode in Vertical Cutting

Fallover is a term used to describe how the tool path reacts to the edge of the cutting surface. If the tool goes beyond the surface or solid edge, it will fall over by some amount. The control of the portion of the tool path behavior is set in the NURBS_XZ and NURBS_YZ cycles by means of a parameter (#28) named Fall Over Mode.

There are 4 possible settings for Fall Over Mode. You may specify 0, 1, 2, 3 for this parameter, which will change the behavior of the tool path as follows:

Setting Fall Over Mode:

Using the Modify Cycle button on the Knowledge Editor, select Parameter #28:

Modify Cycle

Parametric name: ROUTER_XNURBS_CUT-ZIGZAG_XZ

Description	Value
[25]Define Scallop Size(Y/N)	
[26]Vertical Cutting Angle..	0.0
[27]Lace Cutting Paths (Y/N)	Y
[28]Fall Over Mode (0/1/<2>)	3
[29]CW Horz. Cuts.....(Y/N)	
[30]Lead-In Name ("NAME"/N)	
[31]Lead-Out Name ("NAME"/N)	

Edit Value of Selected Parameter

[28]Fall Over Mode (0/1/<2>) ☐

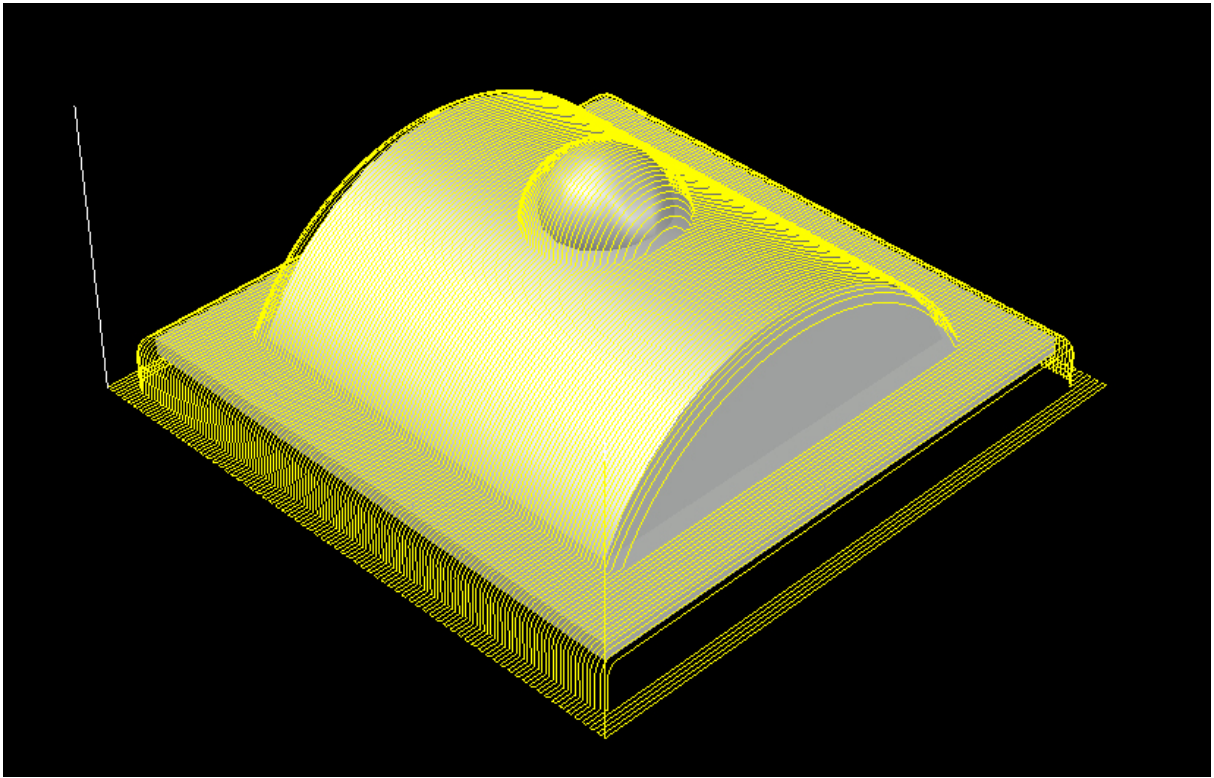
OK Cancel Notes Task Info Write

Permanent Editing

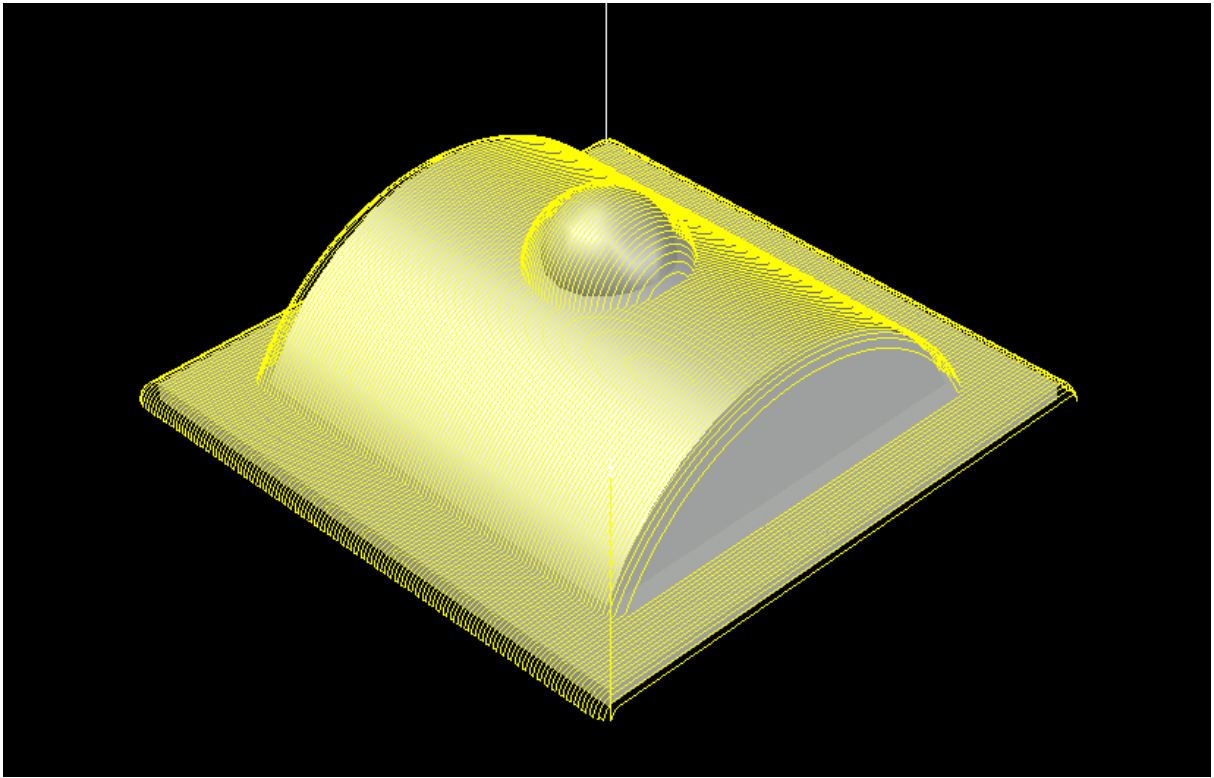
In the Edit Value box, type in the mode you want to set and then press Enter. You should see the list update with the number you typed in.

Fall Over Mode 0:

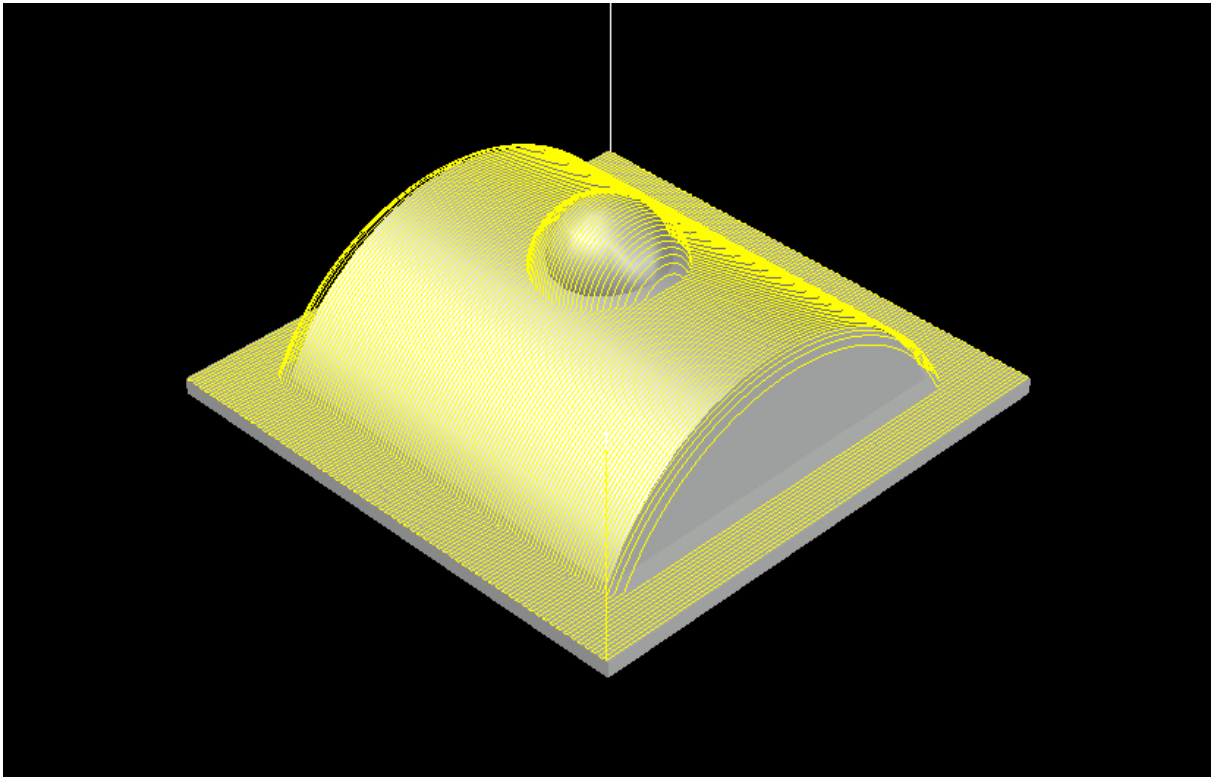
This method will cut down to the Maximum Z Depth set, then move over by a tool diameter in X or Y (depending on cycle) and then move over and return for the next pass.

**Fall Over Mode 1 OR Fall Over Mode 2:**

Fall Over Modes 1 and 2 are the same and switch between vertical slicing in X or Y. There is a slight overcut at the edge of the surface, before the tool path steps over and then returns.

**Fall Over Mode 3:**

This is the default method if none is specified. The tool path will cut to the edge of the solid or surface and stop, move over and return. There is no Fall Over actually done.

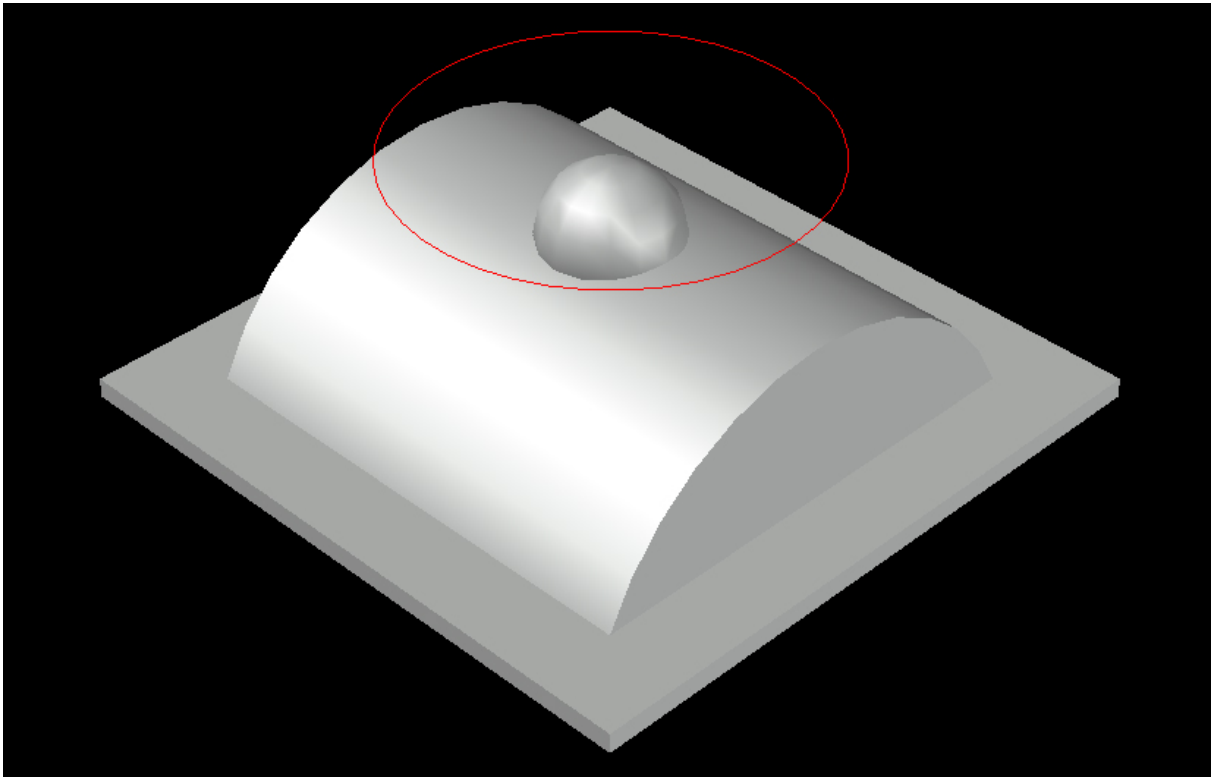


Containment Cutting

Containment cutting is the ability to control the generation of the tool path within a specified area of the part. This is done either by a polyline used as the containment area or by cutting a separate surface if you are drawing with MDT and have NURBS surfaces on your part.

The example given here will show a containment polyline controlling the area where the tool path is created. The polyline must be closed, and actually be a polyline. This is easy to achieve if you are using a circle, ellipse or some other irregular shape or combination of lines and arcs, you can always Geoshape them and use the result from Geoshape for the containment polyline.

In this instance, the MCADTPC.dwg is used with a circular containment area created.



The containment polyline has been created at Z0 and so is above the part.

The cut settings are set up with the Nurbs XZ cycle defaults, but with a Containment Polyline selected in the Containment options.

Cycle Information

Stock Settings

Slice Settings

Rough Parameters

Containment

Recut Settings

Lead Settings

Material Z Location:

Containment

☐ Define Containment Surfaces
☒ Define Containment Polylines

OK

Cancel

Status Information

Safety Plane

Depth Per Pass

Total Cut Depth

Feedrate/Spindle Speed

Feedrate

Spindle Speed

Surface FPM

Units Per Revolution

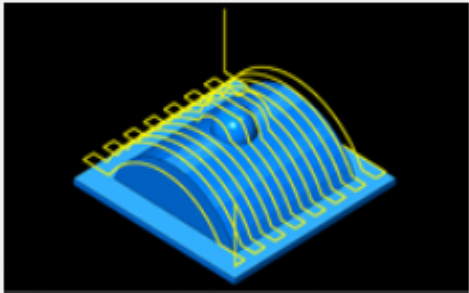
Calculate

Before Codes

After Codes

Oscillation Amount

Sort By Rank #



NURBS XZ

Reset Cycle Settings to Default

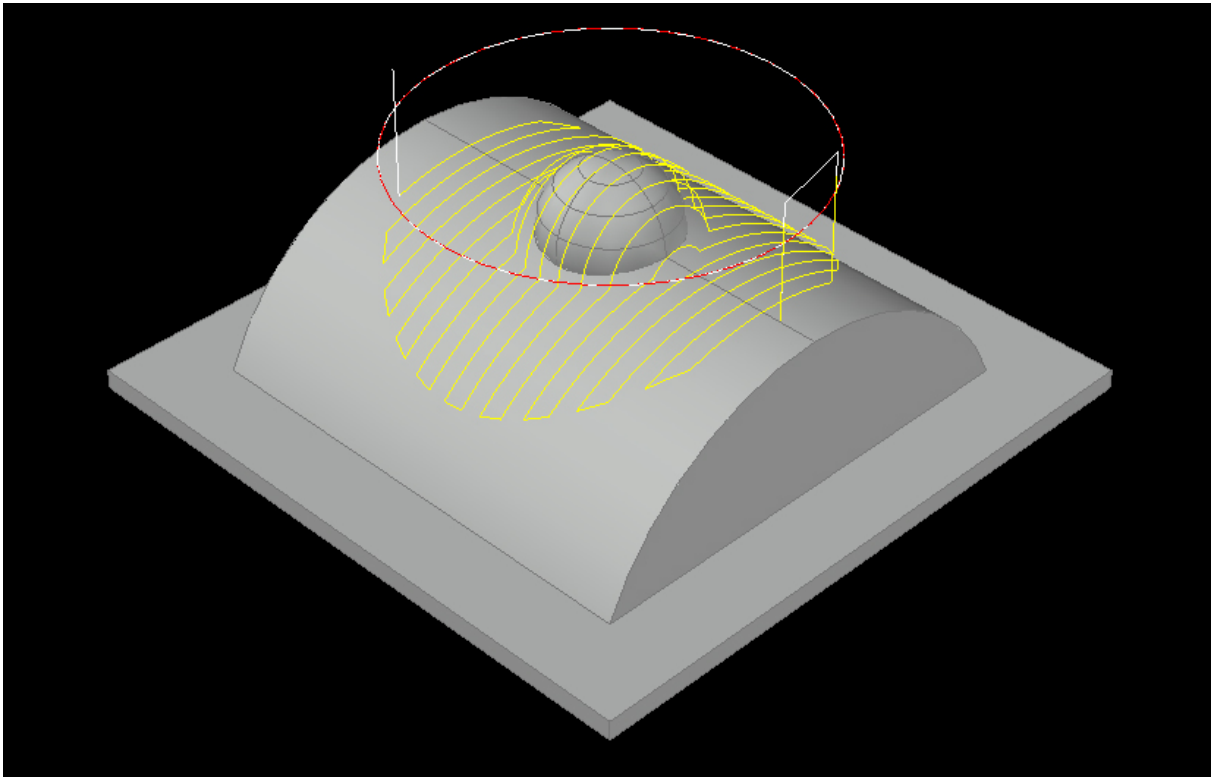
Select Cut, and when prompted to redefine the surfaces, answer Yes. Select the green solid for the part.

You will be prompted to input the Surface Tolerance, use .005.

Enter Surface Tolerance <0.00050000>: .005

Next you will be prompted to select the Containment Polyline(s), select the circular shape above the part.

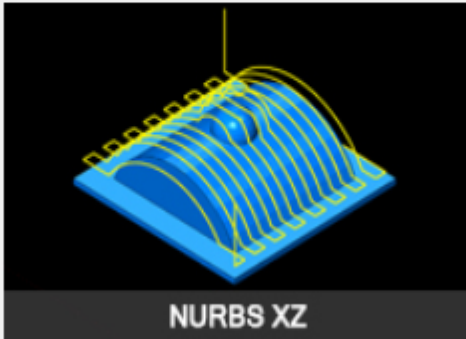
You should see a tool path similar to the one shown here:



If there are any areas where the tool cannot continue its path without crossing the containment line, the tool will be forced up to the Safety Plane and then it will move over to the next area where it can continue the path. This is to avoid gouging the part or violating the containment polyline.

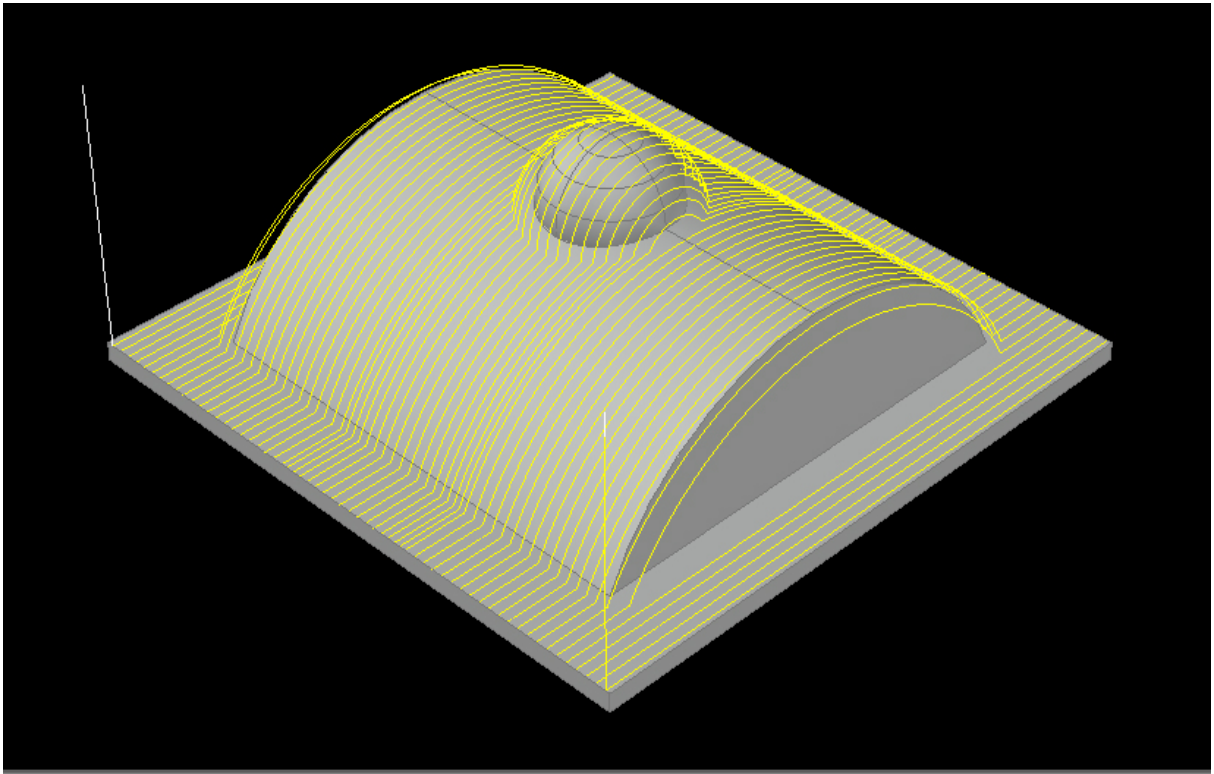
Constant Scallop Height

Scallops are the ridges left behind from the cutter stepping over in X or Y to cut the next pass. A Ball Mill will always leave a small scallop. The size of that scallop can be specified as a height, and Router-CIM will calculate the step over spacing between each pass to maintain that scallop height. Some tool paths will be placed closer together than others, especially if they are by a sloped surface where the tool has to maintain the scallop height while moving up or down as well as in X or Y.

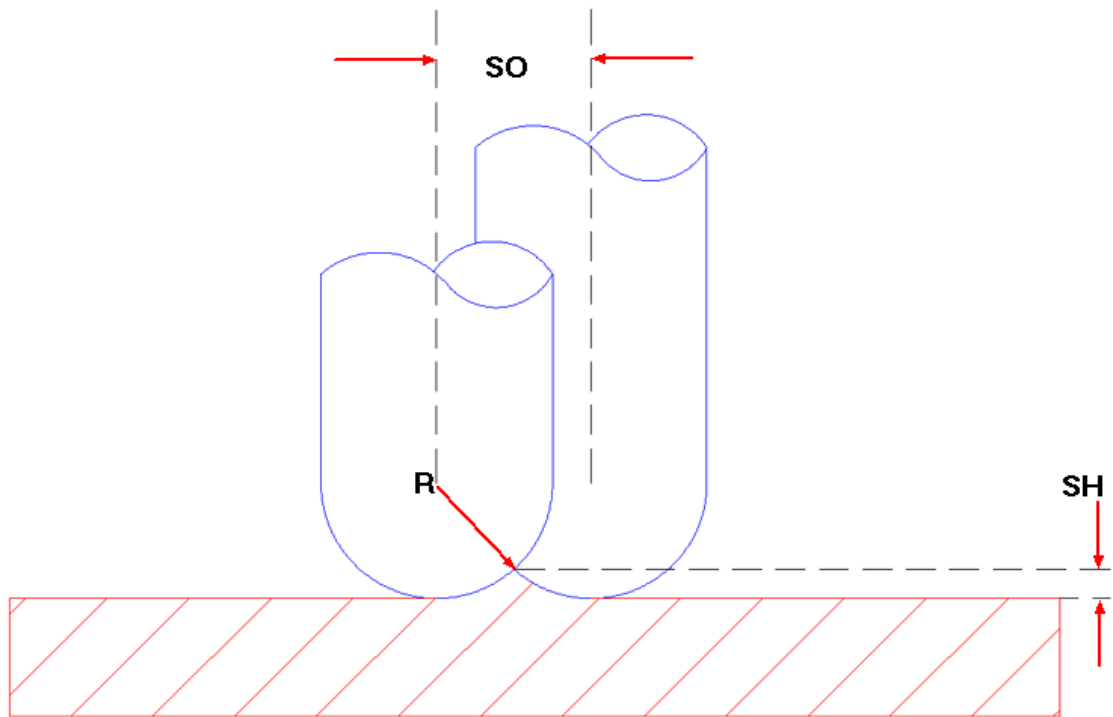
Cycle Information	Status Information
Stock Settings	Safety Plane: *0.25000
Slice Settings	Depth Per Pass: 1.00000
Rough Parameters	Total Cut Depth:
Containment	Feedrate/Spindle Speed
Recut Settings	Feedrate: 1000.00000
Lead Settings	Spindle Speed: 18000.00000
Material Z Location:	Surface FPM: NONE
	Units Per Revolution: NONE
	Calculate
	Before Codes:
	After Codes:
	Oscillation Amount: 0.00000
	Sort By Rank #:
<div> <div>Slice Settings</div> <div> <div>Slice Vertical Angle: 0</div> <div>Slice Spacing: !*tr*</div> <div> <input checked="" type="checkbox"/> Constant Scallop <input checked="" type="checkbox"/> Lace Slices </div> <div> Cut Direction <div> <input type="radio"/> Climb <input checked="" type="radio"/> Conventional </div> </div> <div> Slice Mode <div> <input type="radio"/> ZigZag <input type="radio"/> Unidirectional <input type="radio"/> Closed Spiral </div> </div> <div> <div>OK</div> <div>Cancel</div> </div> </div> </div>	 <div>NURBS XZ</div> <div>Reset Cycle Settings to Default</div>

The below image was created by a 0.005 for Scallop Height and .001 for minimum step size.

These settings will produce a tool path that has varied step over amounts wherever the tool needs to space the tool paths out differently to maintain the scallop height specified. Even though the Slice Spacing was set to !*tr* (Tool Radius), the tool paths are clearly set to an amount very different from that specified so that they can maintain the scallop height of .005". In addition, they are much closer together at the edges of the sphere on top because the tool must step over less in this section to maintain the scallop height.



Router-CIM will maintain the Scallop Height indicated throughout the tool path created. If you want an idea of how much to set the scallop height to or what the step over amount would be, there is a formula to calculate the scallop height:



R = Cutter Radius
 SO = Step Over
 SH = Scallop Height

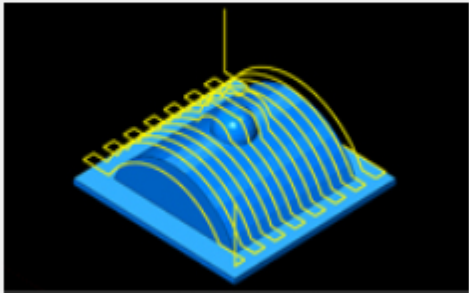
$$SH = R - \sqrt{R^2 - (SO/2)^2}$$

Or another way to express this would be:

$$SH = (R - (\sqrt{R^2 - ((SO/2)^2)}))$$

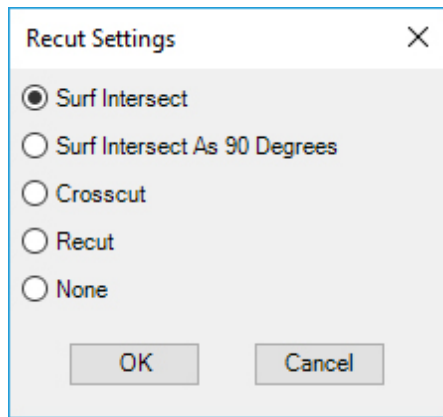
Intersect Cutting

Intersect Cutting will create a separate profile tool path at the intersection of each surface to create a smooth transition at these locations. This can also be useful to remove material in an area where a larger tool was used and left more material on the part than desired.

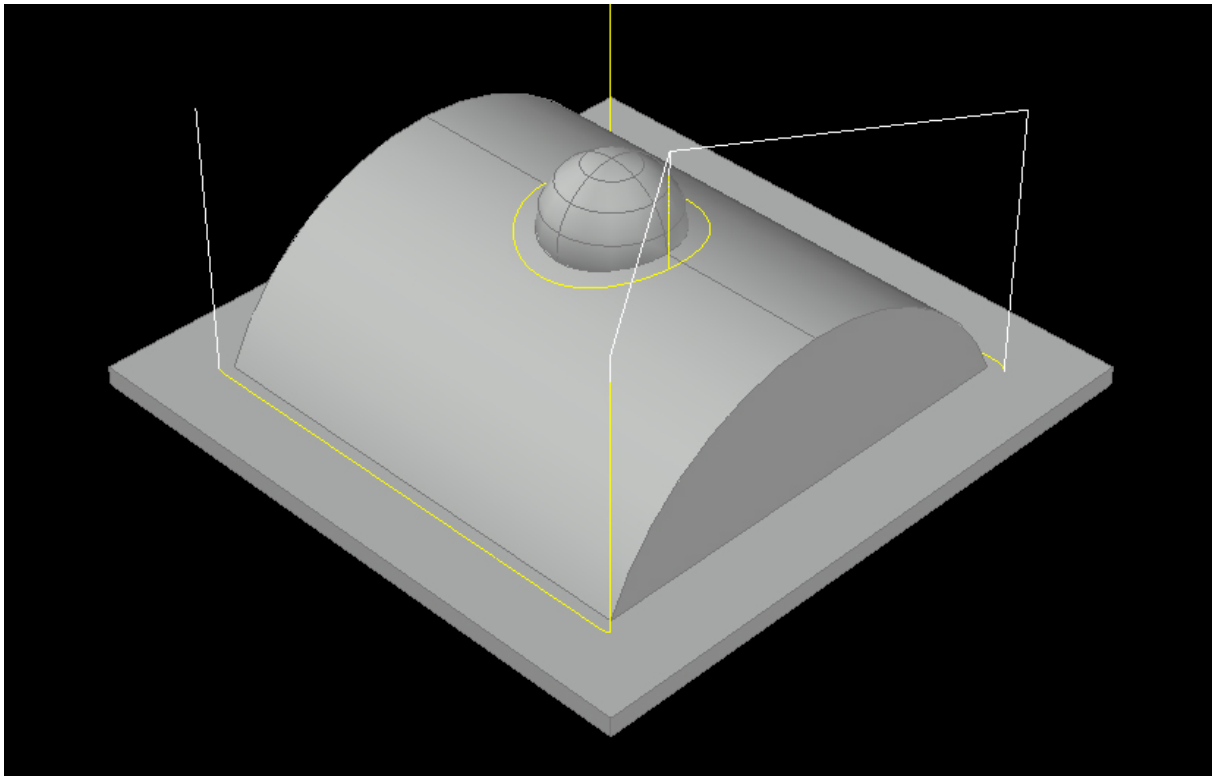
Cycle Information	Status Information
<div>Stock Settings</div>	Safety Plane <input type="text" value="*0.25000"/>
<div>Slice Settings</div>	Depth Per Pass <input type="text" value="1.00000"/>
<div>Rough Parameters</div>	Total Cut Depth <input type="text"/>
<div>Containment</div>	Feedrate/Spindle Speed
<div>Recut Settings</div>	Feedrate <input type="text" value="1000.00000"/>
<div>Lead Settings</div>	Spindle Speed <input type="text" value="18000.00000"/>
Material Z Location: <input type="text"/>	Surface FPM <input type="text" value="NONE"/>
	Units Per Revolution <input type="text" value="NONE"/>
	<div>Calculate</div>
	Before Codes <input type="text"/>
	After Codes <input type="text"/>
	Oscillation Amount <input type="text" value="0.00000"/>
	Sort By Rank # <input type="text"/>
	
	<div>Reset Cycle Settings to Default</div>

Surf Intersect

Shown below is the MCADTPC.dwg with the Surface Intersect turned on in Recut Settings.



Using these settings produces a tool path result similar to this:

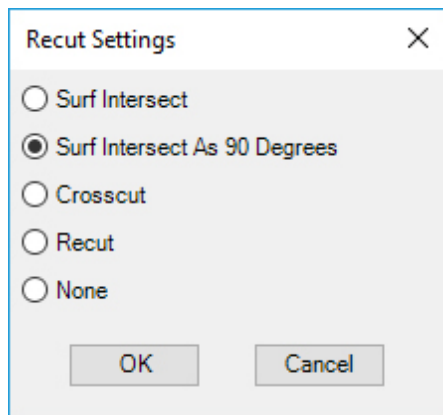


This is a useful feature when a larger tool is used in a roughing or finishing operation and a smaller tool is needed only at the intersections of any geometry to create a smoother transition from one surface to the next.

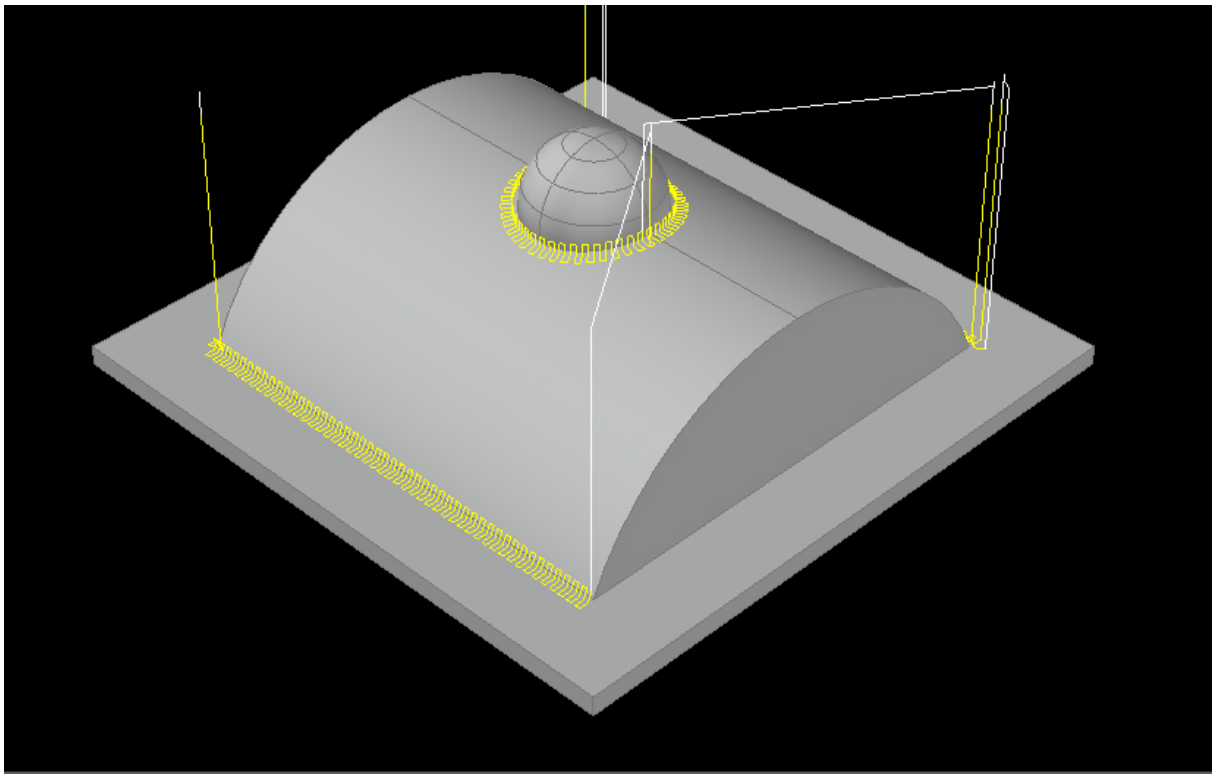
Be aware that this tool path does not consider the radius offset of the tool, so if you are creating this cut with a ball end mill, and the zero point for the end mill is the radius of the tool, then a Stock Allowance equal to the radius of the tool should be used to make sure the tool path is lofted and the part is not overcut.

Surf Intersect at 90 Degrees

Using the Surf Intersect at 90 Degrees creates a Zigzag tool path following the intersection of any of the contours on your part.

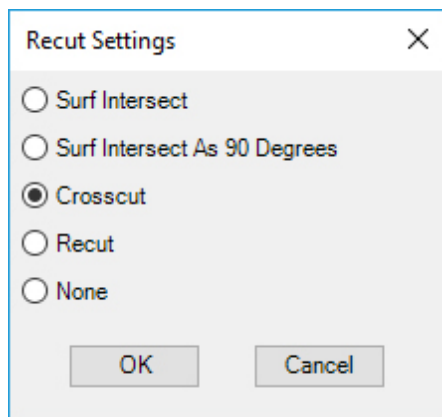


Changing the settings above to use Surf Intersect at 90 Degrees will yield a tool path with these results:

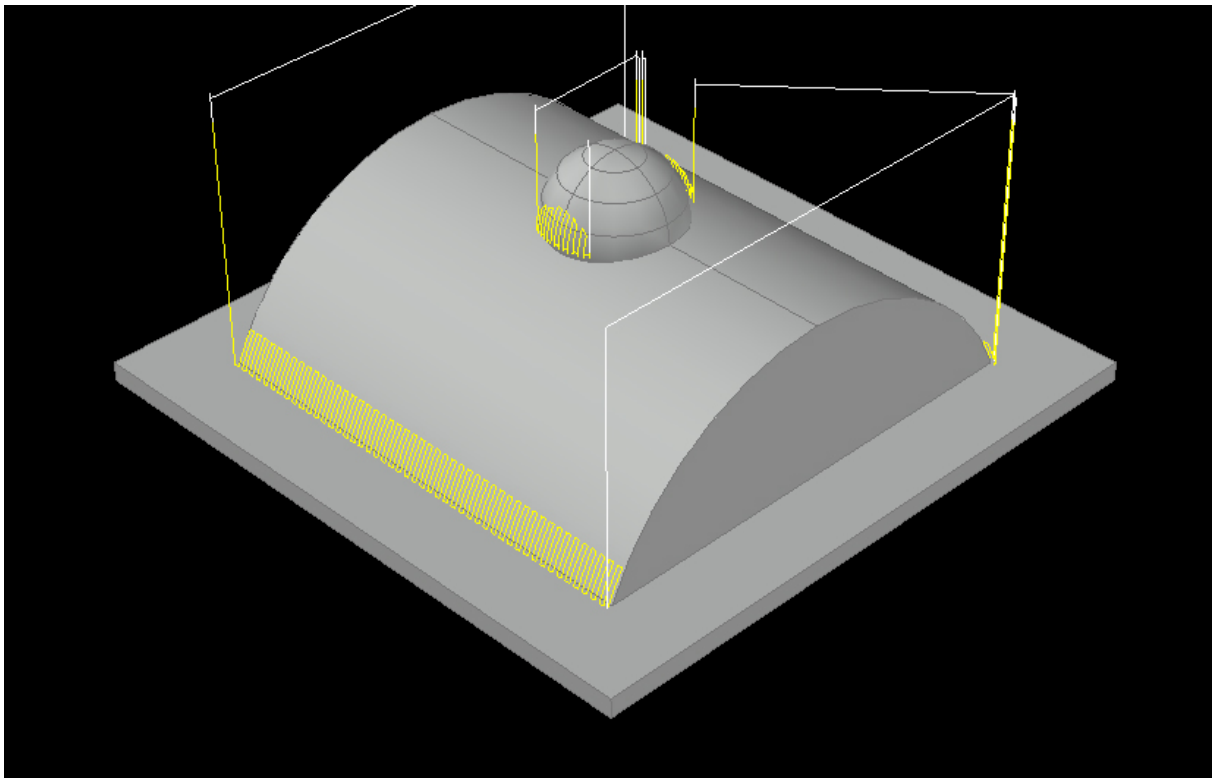


These motions will create many more moves than the first Surf Intersect, resulting in more NC Code, but possibly a smoother transition. The Slice Spacing will usually be changed to yield a much smaller scallop on the part.

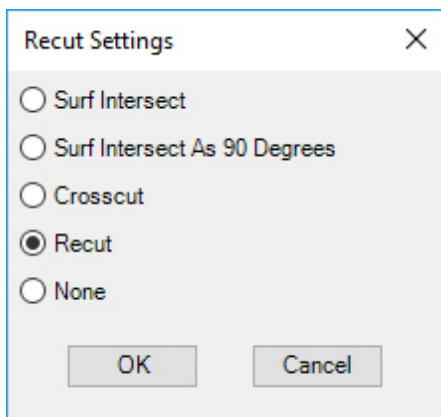
Crosscut



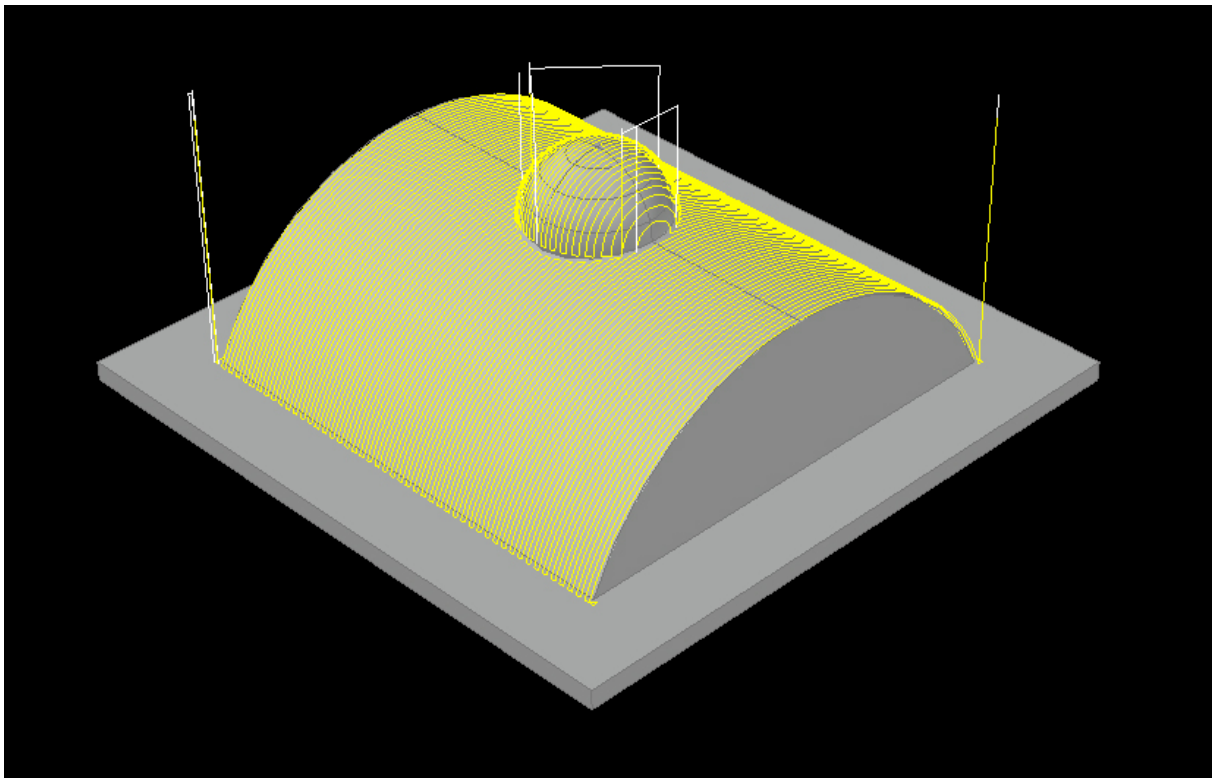
Using these settings produces a tool path result similar to this:



Recut



Using these settings produces a tool path result similar to this:



Roughing

Using roughing passes, you can remove large volumes of material from a part with a large tool prior to making a finish pass (or passes) with a smaller tool. This helps lower the overall cycle time and save on tool wear. There are several methods of rough cutting in Router-CIM, and each will be described in as much detail as possible. There are example cuts shown for each method that employ settings that are intended to show the method and settings of the cycle, though are not necessarily the most efficient settings for that particular tool or cycle.

Roughing Parameters [X]

Rough Z Cut Increment:

Rough Type
☒ 1 ☐ 2 ☐ 3

☐ Plunge Points

Working Area X Minimum:

Working Area Y Minimum:

Working Area Z Minimum:

Working Area X Maximum:

Working Area Y Maximum:

Working Area Z Maximum:

OK Cancel

Vertical Cutting + Mode 1

Offsets each roughing pass by "Rough Z Cut Increment" + "Stock Allowance", moving in rapid when machine motions exceed "Working Area Z Maximum".

The available cycles for this method are **Nurbs_XZ** and **Nurbs_YZ**.

Vertical Cutting + Mode 2

Offsets each roughing pass by "Rough Z Cut Increment" + "Stock Allowance", and proportionally distributes Z depth throughout the roughing activity without allowing tool to leave material.

The available cycles for this method are **Nurbs_XZ** and **Nurbs_YZ**.

Vertical Cutting + Mode 3

Shifts roughing motions in "Z" based on "Rough Z Cut Increment" while compensating for "Stock Allowance", moving in rapid when machine motions exceed "Working Area Z Maximum".

The available cycles for this method are **Nurbs_XZ** and **Nurbs_YZ**.

Horizontal Cutting + Mode 1

Plunges straight down to each Z level, compensates for "Stock Allowance", roughs without following collective surface contours before proceeding to next lower Z level. Plunge Points can be used in conjunction with this method of roughing.

The available cycle for this methods is **Nurbs_CCW_XY**.

Horizontal Cutting + Mode 2

Ramps down to each Z level, compensates for "Stock Allowance", roughs and makes a contour following pass while ramping down to next Z level.

The available cycle for this methods is **Nurbs_CCW_XY**.

Horizontal Cutting + Mode 3

Plunges straight down to each Z level, compensates for "Stock Allowance", roughs and makes a contour following pass before plunging straight down to next Z level. Plunge Points can be used in conjunction with this method of roughing.

The available cycle for this methods is **Nurbs_CCW_XY**.

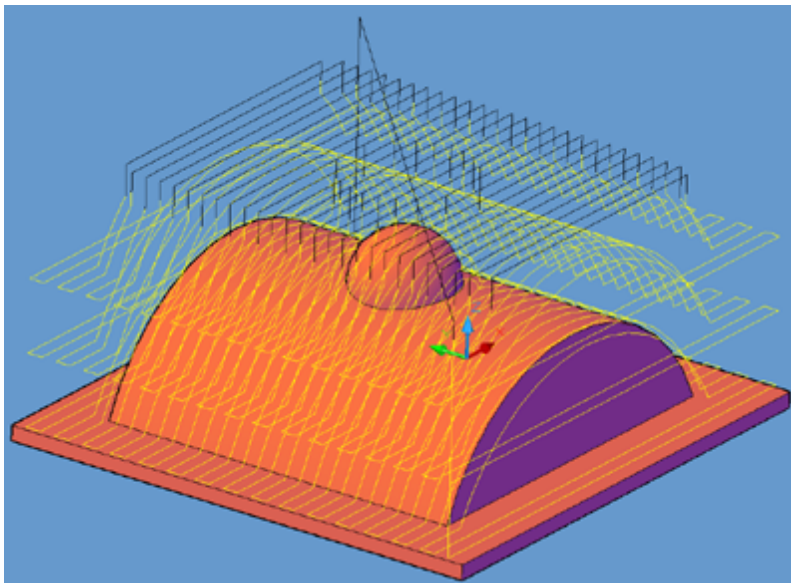
Vertical Roughing Mode 1

Vertical Roughing Mode 1 creates offsets of the shape at the various Z levels, and removes material where it exists at that Z level, indexing to the next location where there is no material to remove. The basic idea of this cycle is that the tool will be creating an approximation of the shape at each level, where the shape exists, and index where it does not to save time, and not cut 'air'.

This method is efficient if the tool can load and unload during the cut, and the indexing during some of the passes is not prohibitive to the tool or part.

In this case the first pass removes some material from the sides of the block, but spends some time indexing to the next cut position, before dropping down to the second level. The second level keeps the tool in the material for all but the top of the dome, and the third level keeps the tool in the material the entire time for the last rough pass.

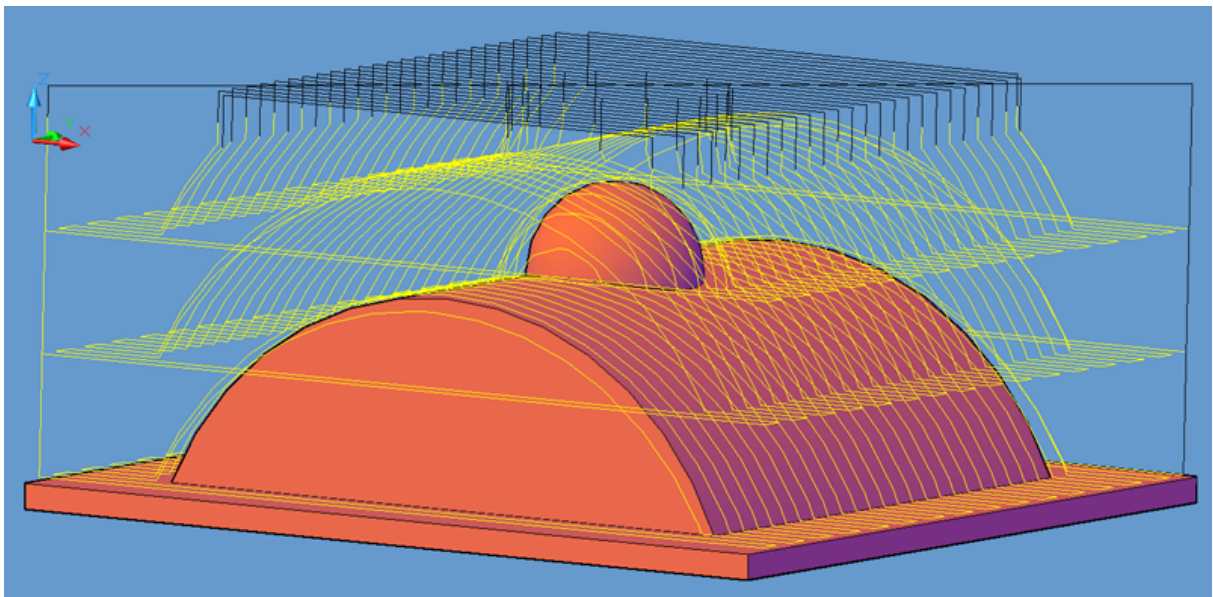
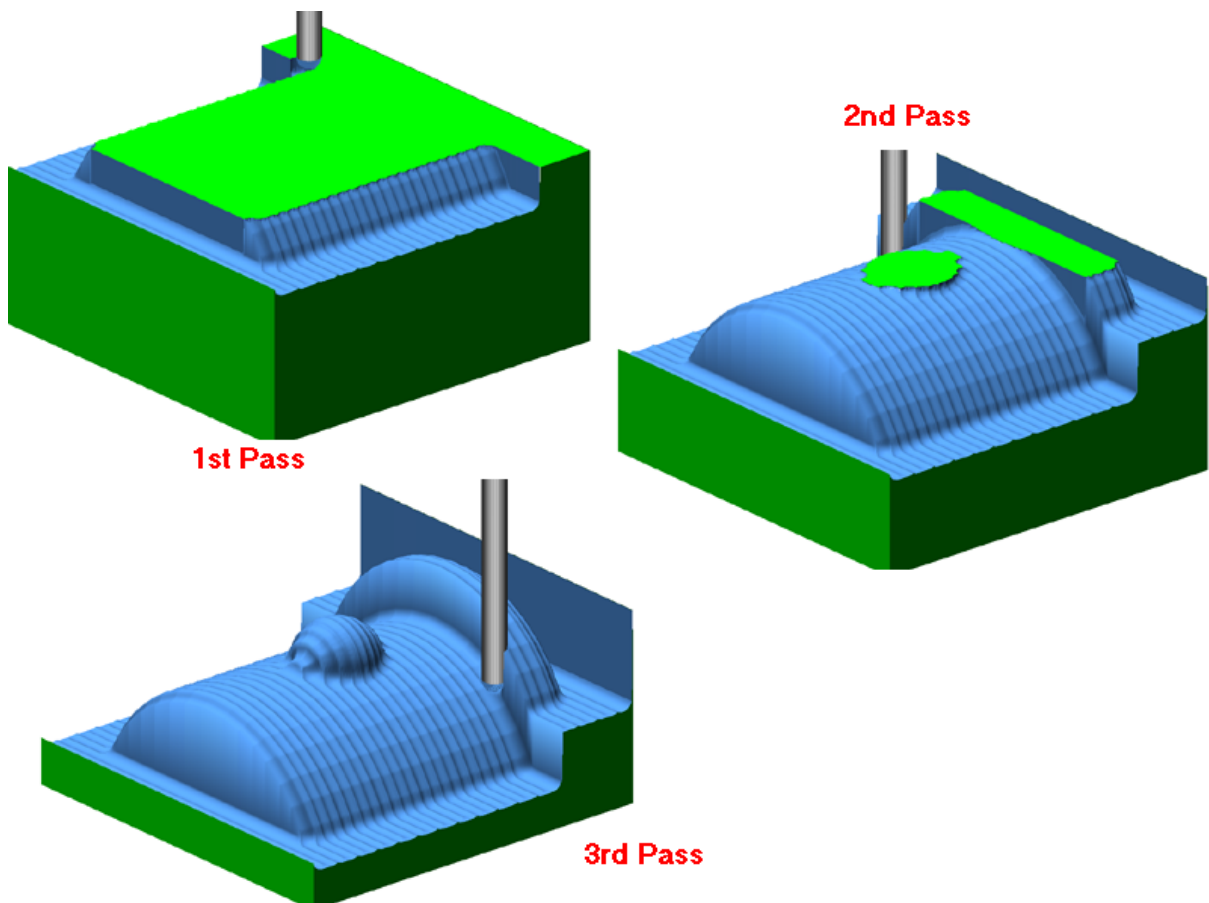
These settings can be used with either the **Zigzag XZ** or **Zigzag YZ** cycles.



Using the MCADTPC.dwg and the following parameters, a similar roughing pass can be created.

The screenshot displays the Router-CIM software interface with the following components:

- Cycle Information Panel:** A vertical stack of buttons: Stock Settings, Slice Settings, **Rough Parameters** (highlighted with a blue border), Containment, Recut Settings, and Lead Settings. Below these is a text field for "Material Z Location:".
- Status Information Panel:** A list of parameters with input fields:
 - Safety Plane: *0.25000
 - Depth Per Pass: 1.00000
 - Total Cut Depth: (empty)
 - Feedrate/Spindle Speed section:
 - Feedrate: 1000.00000
 - Spindle Speed: 18000.00000
 - Surface FPM: NONE
- Stock Settings Dialog:** A modal window with a close button (X). It contains:
 - Stock Allowance: 0.0625
 - ☐ Vary Stock Allowance
 - OK and Cancel buttons.
- Roughing Parameters Dialog:** A modal window with a close button (X). It contains:
 - Rough Z Cut Increment: 0.75
 - Rough Type: Radio buttons for 1 (selected), 2, and 3.
 - ☐ Plunge Points
 - Working Area X Minimum, Y Minimum, Z Minimum, X Maximum, Y Maximum, and Z Maximum fields.
 - OK and Cancel buttons.



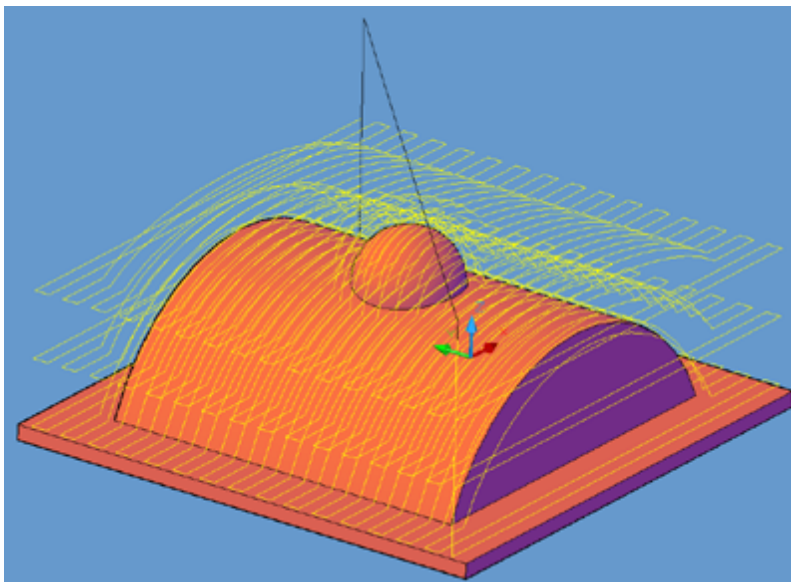
top few passes would index at the Safety Plane for some of their cuts, but the bottom passes would constantly be in material.

Vertical Roughing Mode 2

Vertical Roughing Mode 2 will remove material constantly from the part on each pass, with each pass approximating the part at that level. With this method the tool stays loaded during each pass, on each Z level until the cut is finished. This cycle cuts in a Zigzag fashion, so at the end of a pass the tool moves over and starts cutting back on the next pass, always in material.

This method of roughing is efficient when the tool must stay loaded up during the cut to get the most satisfactory performance. Since the tool is always cutting and does not index during any of the passes, this is the longest roughing cycle in terms of machine cycle time.

In this instance, the first pass removes material from the block, and cuts part of a contoured shape resembling the part at this Z level. The second pass removes more material leaving a more distinct impression of the part and still keeping the tool loaded during the entire pass. The last Z level finishes the roughing and removes the same volume of material as the previous passes.



Using the MCADTPC.dwg and the following parameters, a similar roughing pass can be created.

Cycle Information

Stock Settings

Slice Settings

Rough Parameters

Containment

Recut Settings

Lead Settings

Material Z Location:

Stock Settings

Stock Allowance:

☐ Vary Stock Allowance

OK Cancel

Status Information

Safety Plane

Depth Per Pass

Total Cut Depth

Feedrate/Spindle Speed

Feedrate

Spindle Speed

Surface FPM

Roughing Parameters

Rough Z Cut Increment:

Rough Type

☐ 1 ☒ 2 ☐ 3

☐ Plunge Points

Working Area X Minimum:

Working Area Y Minimum:

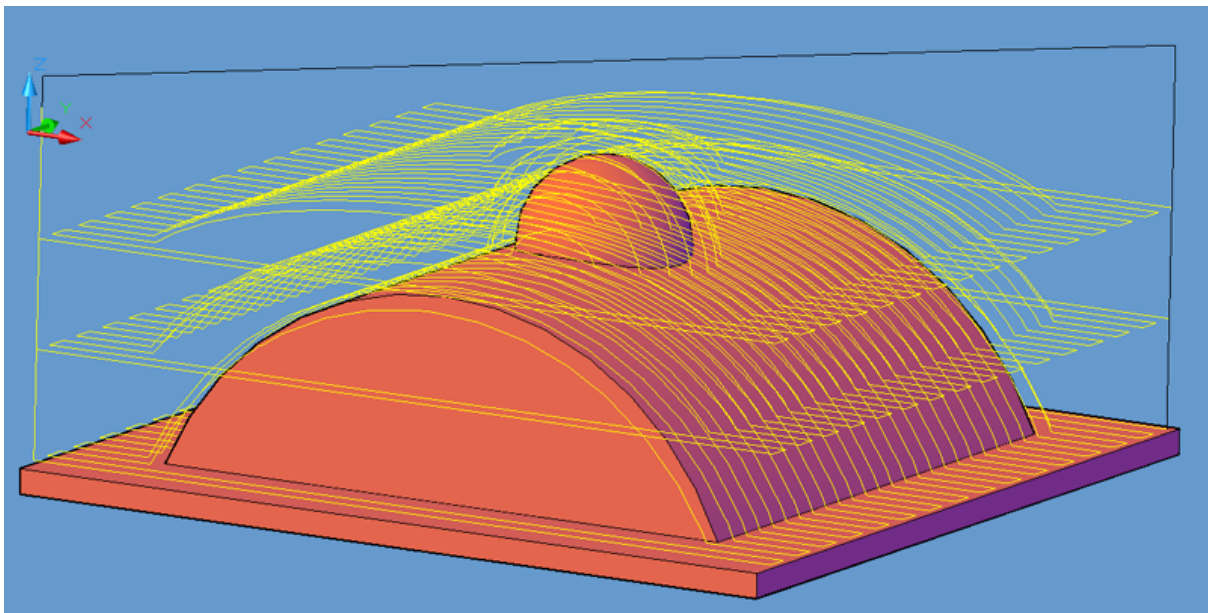
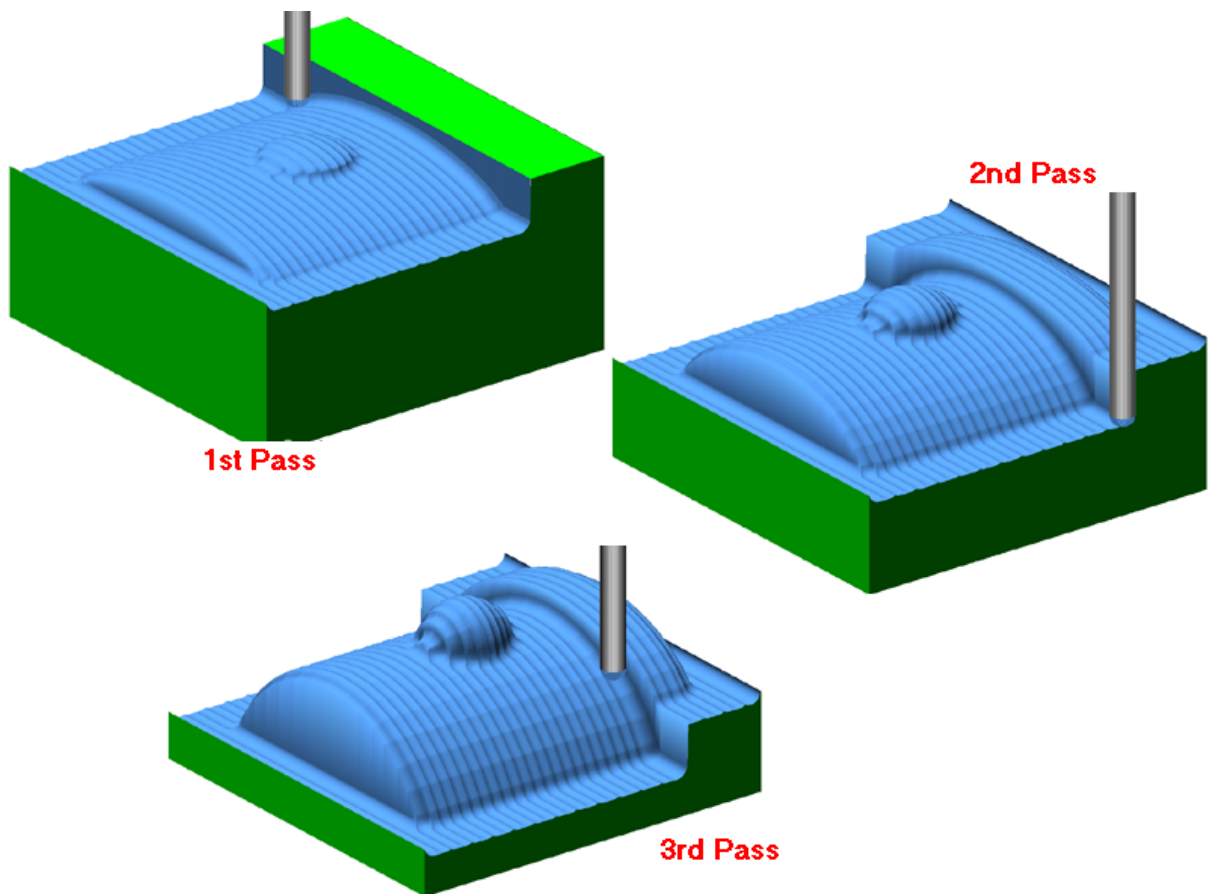
Working Area Z Minimum:

Working Area X Maximum:

Working Area Y Maximum:

Working Area Z Maximum:

OK Cancel

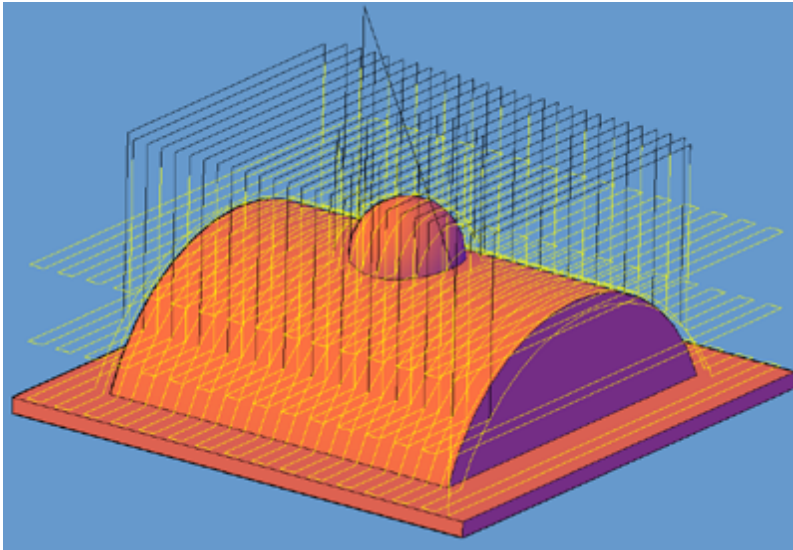


These rouging tool paths show how the tool stays in the material cutting, never indexing during any of the passes.

Vertical Rouging Mode 3

Vertical Rouging Mode 3 will remove material at the plane of the cut, and remove material from the part at the finished roughing size on each pass. Router-CIM will cause the tool to move to the Safety Plane and index to the next position if there is no material to remove from the part, or the block of material on the current pass. In the example that follows, the first pass is made removing material from the block and only the ball section at the top of the part. That ball section is never cut again on any of the subsequent passes, as the first pass removed all the material necessary. The second pass removes some material from the side of the part and indexes up to the Safety Plane and over to the other side of the part then down into the material again when there is material to remove from the block. The last pass removes material from the bottom section of the part but does not cut any of the part that was already cut on the first or second pass.

This is an efficient method of removing material if the tool can load/unload during the cut without affecting the part or the tool.



Using MCADTPC.dwg, and the following settings, a similar rouging tool path can be created.

The screenshot displays the Router-CIM software interface. The main window is divided into two panes. The left pane, titled "Cycle Information", contains a list of settings: Stock Settings, Slice Settings, Rough Parameters (highlighted with a blue border), Containment, Recut Settings, and Lead Settings. Below this list is a "Material Z Location:" label and an empty text box. The right pane, titled "Status Information", displays various parameters: Safety Plane (*0.25000), Depth Per Pass (1.00000), Total Cut Depth (empty), Feedrate/Spindle Speed (1000.00000), Spindle Speed (18000.00000), and Surface FPM (NONE). Two modal dialog boxes are open. The "Stock Settings" dialog box, in the foreground, has a "Stock Allowance:" field set to 0.0625 and a "Vary Stock Allowance" checkbox. The "Roughing Parameters" dialog box, in the background, has a "Rough Z Cut Increment:" field set to 0.75, a "Rough Type" section with radio buttons 1, 2, and 3 (where 3 is selected), a "Plunge Points" checkbox, and several "Working Area" fields (X Minimum, Y Minimum, Z Minimum, X Maximum, Y Maximum, Z Maximum) with values 0, 0, 0, 0, 0, and 0 respectively. Both dialog boxes have "OK" and "Cancel" buttons.

Cycle Information

Stock Settings

Slice Settings

Rough Parameters

Containment

Recut Settings

Lead Settings

Material Z Location:

Stock Settings

Stock Allowance: 0.0625

☐ Vary Stock Allowance

OK Cancel

Status Information

Safety Plane *0.25000

Depth Per Pass 1.00000

Total Cut Depth

Feedrate/Spindle Speed

Feedrate 1000.00000

Spindle Speed 18000.00000

Surface FPM NONE

Roughing Parameters

Rough Z Cut Increment: 0.75

Rough Type

☐ 1 ☐ 2 ☒ 3

☐ Plunge Points

Working Area X Minimum:

Working Area Y Minimum:

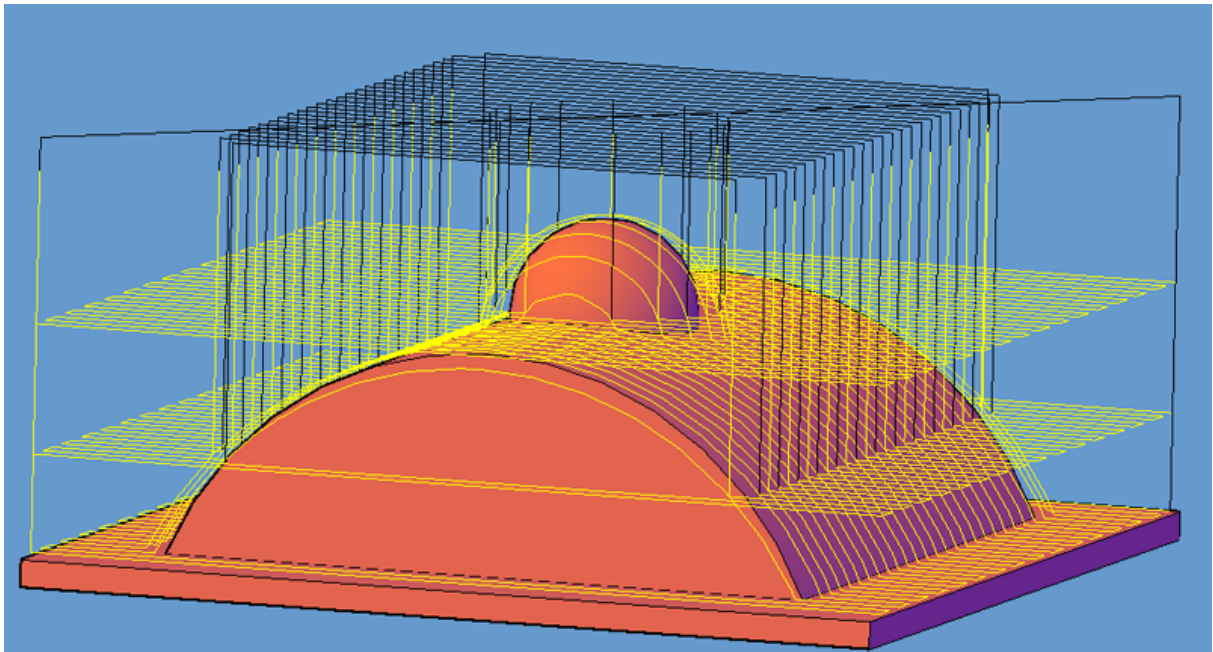
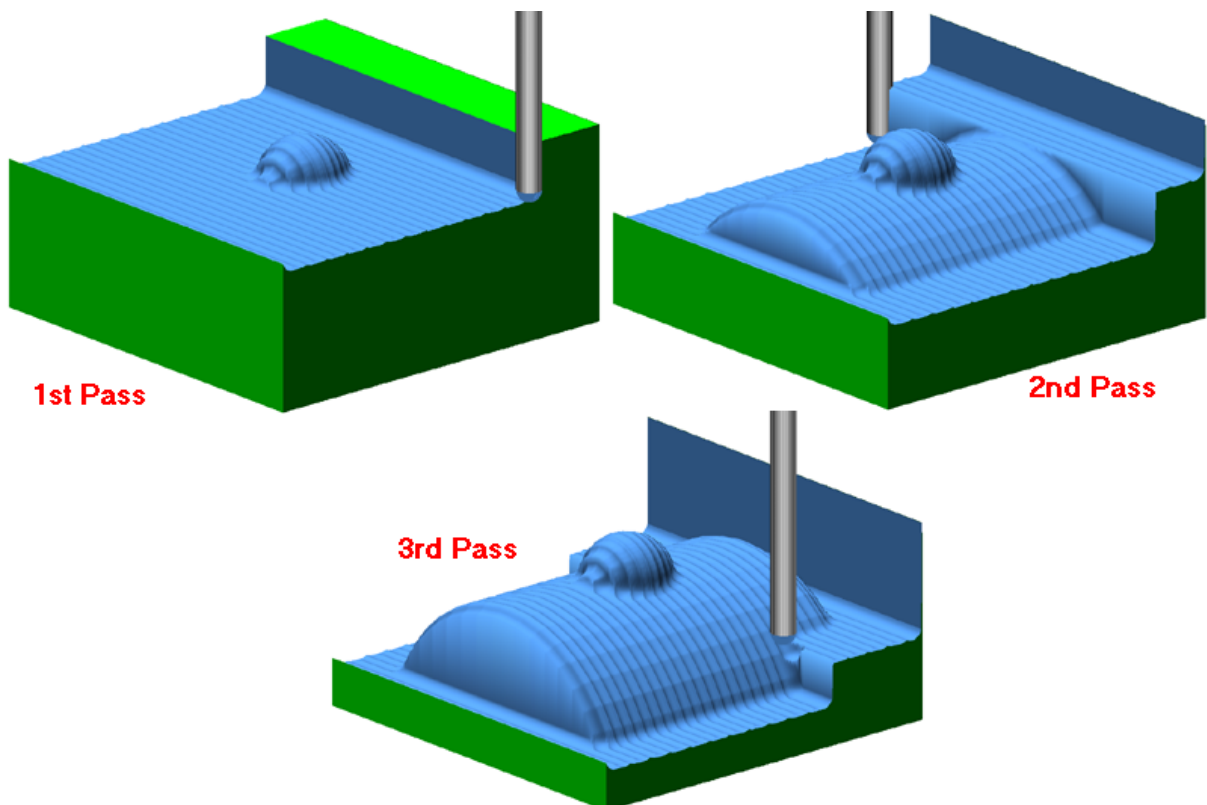
Working Area Z Minimum:

Working Area X Maximum:

Working Area Y Maximum:

Working Area Z Maximum: 0

OK Cancel



These roughing passes show where the tool lifts up and indexes over to the next cut position, only removing material from the part itself once at each particular Z level.

Examples

To begin we will use the "MILL_XNURBS_CUT-ZIGZAG_XZ" Cycle. Proper operation of the Expert NURBS Cutter requires the establishment of a relationship between the surfaces to be cut, and the top of the stock material. The surfaces in the example parts MCADTPC.DWG and MCADFLOW.DWG, have been positioned as though the top of the stock material is located at Z 0.0. All surfaces are positioned below Z 0.0.

This is an important step as unreliable or confusing results can occur with geometry above Z0.0 in the cad drawing.

Note: MCADTPC.DWG and MCADFLOW.DWG were included with the installation of Router-CIM and can be found in C:\rcim-work folder.

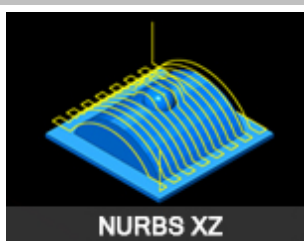
A ".GEO" file is produced by the system the first time you "Cut" the surfaces contained in the example drawings. When following along in the examples, always respond YES to the "Define Surfaces" prompt, because there are times when the original ".GEO" file is inappropriate for the exercise in the example.

Each Example will include screen captures of the Knowledge Editor, containing the proper entries to produce the results shown in the example. The only time you will be required to use the Edit Cycle function is in the Vertical Cutting Fallover Example #5. Fallover Mode is managed by the system for non finishing operations. The default Fallover Mode is Mode 3. Fallover Mode settings do not normally change on a day to day basis. Therefore, Fallover is not in the Knowledge Editor GUI to keep the operation of the system as simple as possible.

Detailed information on each of the Cycle Parameters is found in the previous section under [NURBS CYCLE PARAMETERS](#).

The purpose of the Examples is to illustrate a variety of toolpath results, by varying the Knowledge Editor, or Cycle Parameter entries. Feed and Speed Settings do not effect the graphic results of toolpath.

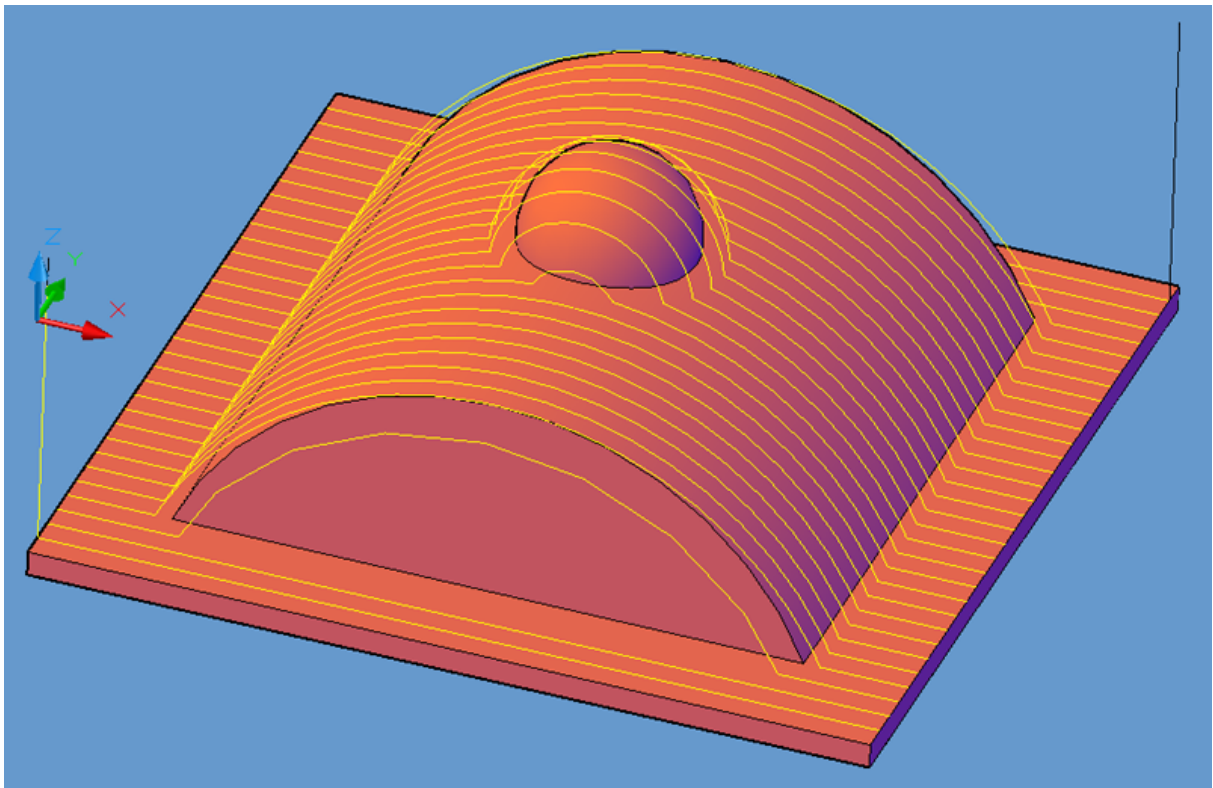
NURBS XZ



The NURBS XZ cycle is a Vertical Slicing cycle for cutting solids and surfaces inside of AutoCAD. Tool paths will be generated in the X direction as slices following the contour of the shape in Z. The slices can be made in one direction, or laced together to allow slicing in both X directions (positive and negative), forming a zig-zag motion of the tool.

Essentially the NURBS XZ cycle is the same as the NURBS YZ cycle with a Slicing Angle of 0.0°.

There are many parameters available to fine tune this cycle to perform many operations and these will be described throughout the NURBS section of this document.



Shown is a tool path created with the default settings of the NURBS XZ cycle. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

```
Enter Surface Tolerance <0.00050000>:
```

```
Enter .003
```

```
Enter Surface Tolerance <0.00050000>: .003
```

You should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Vertical Finishing at Angle 0.0

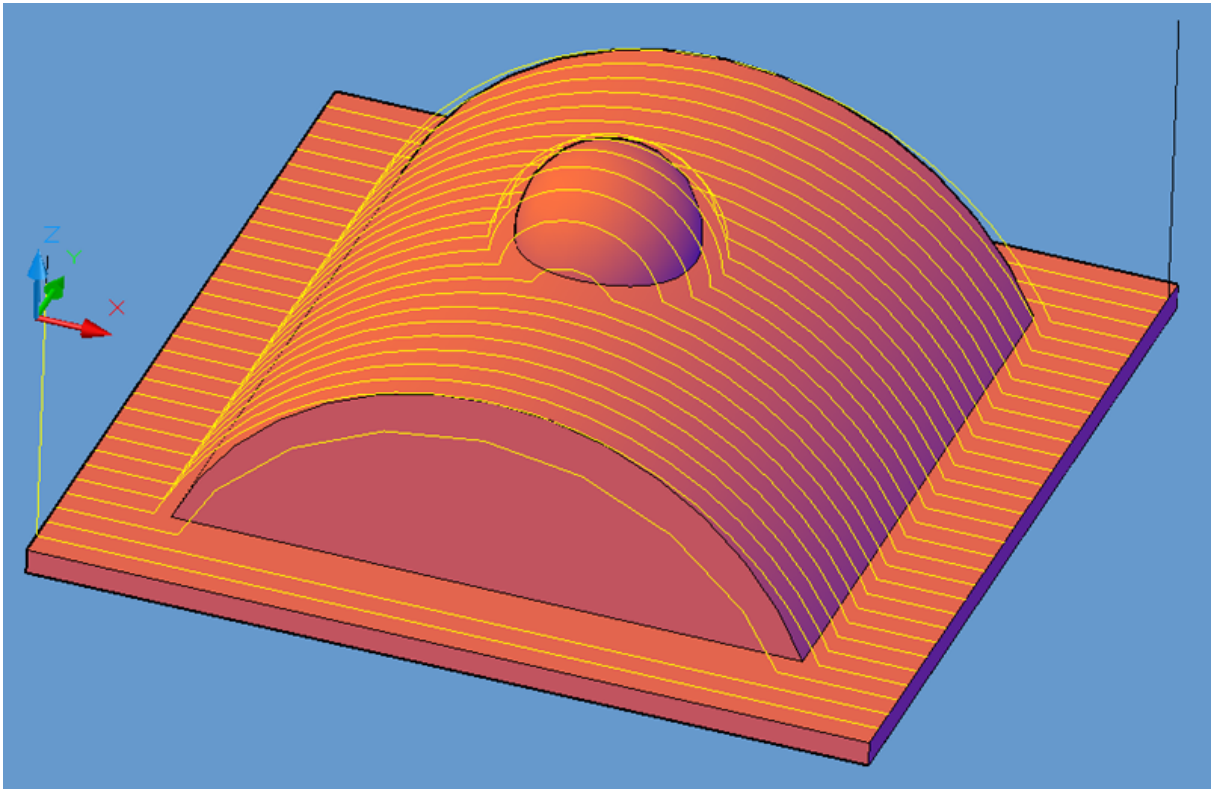
Using the MCADTPC.dwg file, select the XNURBS_CUT-ZIGZAG_XZ Cycle.

Select Slice Settings to be sure the Slice Vertical Angle is set to 0.0.

The screenshot displays the Router-CIM software interface. On the left, the 'Cycle Information' panel has several tabs: 'Stock Settings', 'Slice Settings' (highlighted with a blue border), 'Rough Parameters', 'Containment', 'Recut Settings', and 'Lead Settings'. Below these tabs is a 'Material Z Location' input field. A 'Slice Settings' dialog box is open in the foreground, showing the following options: 'Slice Vertical Angle' set to 0.0, 'Slice Spacing' set to 1.0, 'Constant Scallop' unchecked, 'Lace Slices' checked, 'Cut Direction' set to 'Conventional' (radio button selected), and 'Slice Mode' set to 'ZigZag' (radio button selected). The 'OK' and 'Cancel' buttons are at the bottom of the dialog. On the right, the 'Status Information' panel contains various input fields: 'Safety Plane' (0.25000), 'Depth Per Pass' (1.00000), 'Total Cut Depth' (empty), 'Feedrate/Spindle Speed' section with 'Feedrate' (1000.00000), 'Spindle Speed' (18000.00000), 'Surface FPM' (NONE), and 'Units Per Revolution' (NONE). A 'Calculate' button is below these fields. Further down are 'Before Codes', 'After Codes', 'Oscillation Amount' (0.00000), and 'Sort By Rank #' (empty). At the bottom right is a 3D visualization of a tool path on a blue surface, labeled 'NURBS XZ'. Below the visualization is a 'Reset Cycle Settings to Default' button.

Press the '**Cut**' Button.

The output of these settings should produce a tool path like this:



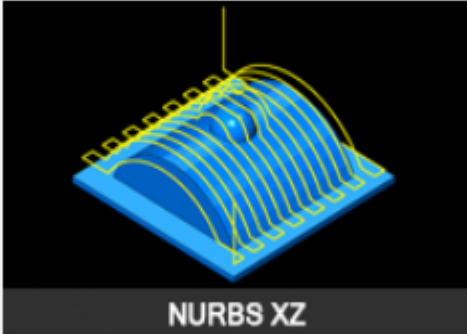
The settings used are finishing settings. There is no Stock Allowance used. There are no roughing passes, lead-in, or lead-out. Also Containment and Recutting are turned off. Each of those setting will be explained, just a little bit later in this document.

The main purpose here is that you understand how the XZ slice creates a basic tool path in the X and Z direction with step over in the Y.

Vertical Finishing at Angle Minus 45

Using the MCADTPC.dwg file, select the XNURBS_CUT-ZIGZAG_XZ Cycle.

Select Slice Settings to be sure the Slice Vertical Angle is set to -45.

Cycle Information		Status Information	
Stock Settings		Safety Plane	*0.25000
Slice Settings		Depth Per Pass	1.00000
Rough Parameters		Total Cut Depth	
Containment		Feedrate/Spindle Speed	
Recut Settings		Feedrate	1000.00000
Lead Settings		Spindle Speed	18000.00000
Material Z Location:		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		NURBS XZ	
		Reset Cycle Settings to Default	

Slice Settings

Slice Vertical Angle:

-45

Slice Spacing:

!tr*

☐ Constant Scallop

☒ Lace Slices

Cut Direction

☐ Climb

☒ Conventional

Slice Mode

☐ ZigZag

☐ Unidirectional

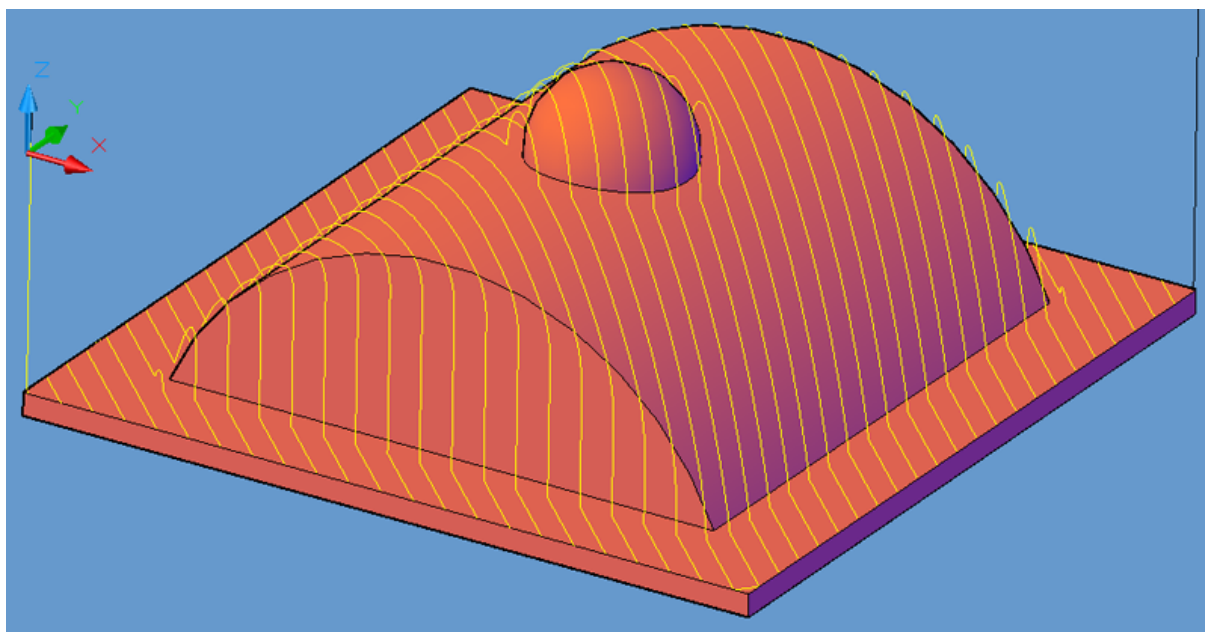
☐ Closed Spiral

OK

Cancel

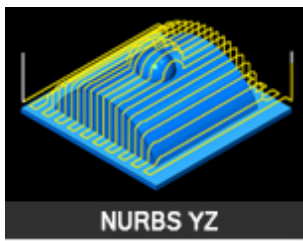
Press the **'Cut'** Button.

These settings should produce a tool path like this:



The only difference between this tool path and Vertical Finishing at Angle 0.0 is the setting for Slice Vertical Angle. Setting this to a value other than 0 produces slices on the specified angle.

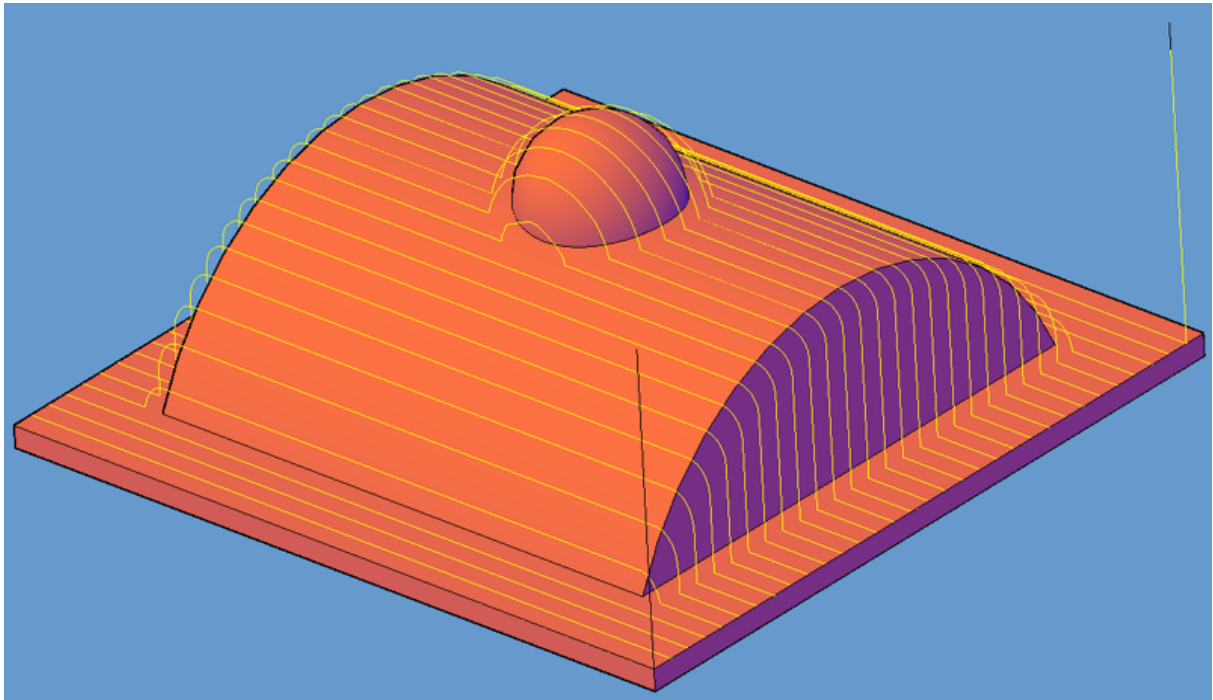
NURBS YZ



The NURBS YZ cycle is a Vertical Slicing cycle for cutting solids and surfaces inside of AutoCAD. Tool paths will be generated in the Y direction as slices following the contour of the shape in Z. The slices can be made in one direction, or laced together to allow slicing in both X directions (positive and negative), forming a zig-zag motion of the tool.

Essentially the NURBS YZ cycle is the same as the NURBS XZ cycle with a Slicing Angle of 90°.

There are many parameters available to fine tune this cycle to perform many operations and these will be described throughout the NURBS section of this document.



Shown is a tool path created with the default settings of the NURBS YZ cycle. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

```
Enter Surface Tolerance <0.00050000>:
```

```
Enter .003
```

```
Enter Surface Tolerance <0.00050000>: .003
```

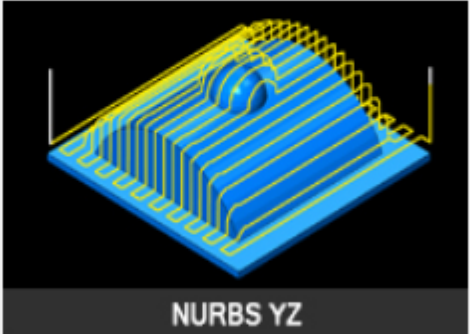
You should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Vertical Finishing in YZ

Using the MCADTPC.dwg file, select the XNURBS_CUT-ZIGZAG_YZ Cycle.

Select Slice Settings to be sure the Slice Vertical Angle is set to 90.0.

Cycle Information		Status Information	
Stock Settings		Safety Plane	*0.25000
Slice Settings		Depth Per Pass	1.00000
Rough Parameters		Total Cut Depth	
Containment		Feedrate/Spindle Speed	
Recut Settings		Feedrate	1000.00000
Lead Settings		Spindle Speed	18000.00000
Material Z Location:		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Slice Settings

Slice Vertical Angle:

90

Slice Spacing:

!tr*

☐ Constant Scallop

☒ Lace Slices

Cut Direction

☐ Climb ☒ Conventional

Slice Mode

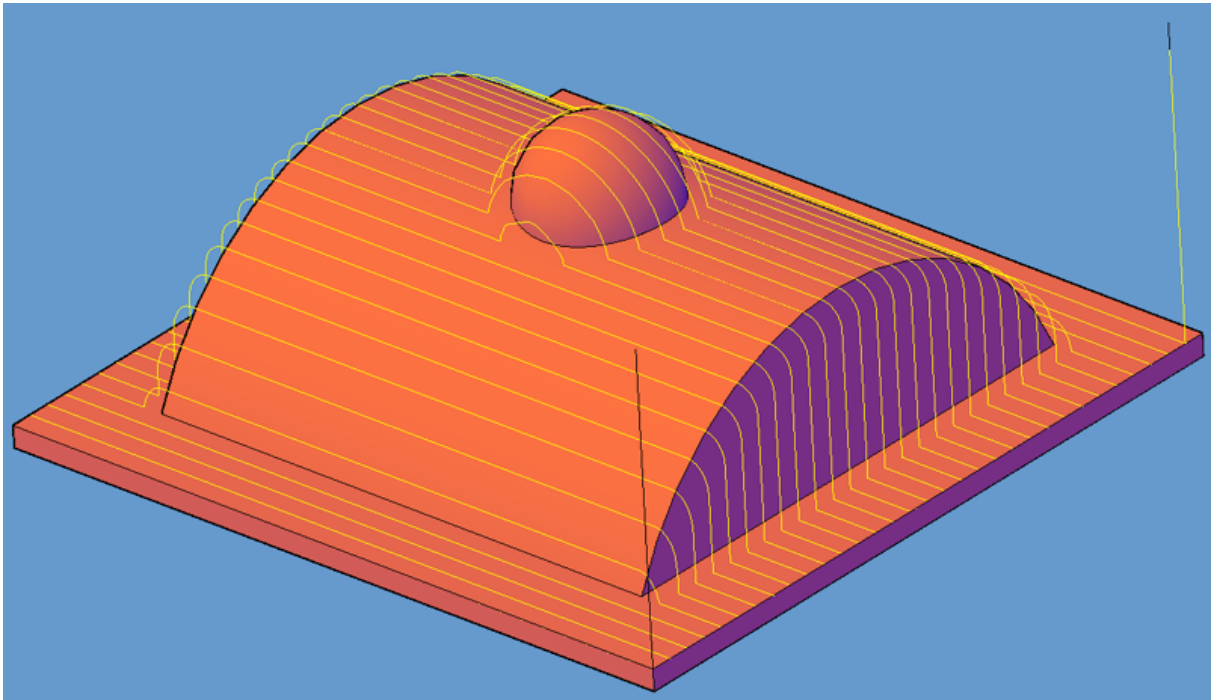
☐ ZigZag ☐ Unidirectional ☐ Closed Spiral

OK

Cancel

Press the **'Cut'** Button.

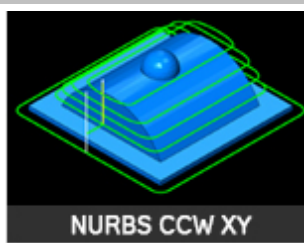
The output of these settings should produce a tool path like this:



The settings used are finishing settings. There is no Stock Allowance used. There are no roughing passes, lead-in, or lead-out. Also Containment and Recutting are turned off. Each of those setting will be explained, just a little bit later in this document.

The main purpose here is that you understand how the YZ slice creates a basic tool path in the Y and Z direction with step over in X.

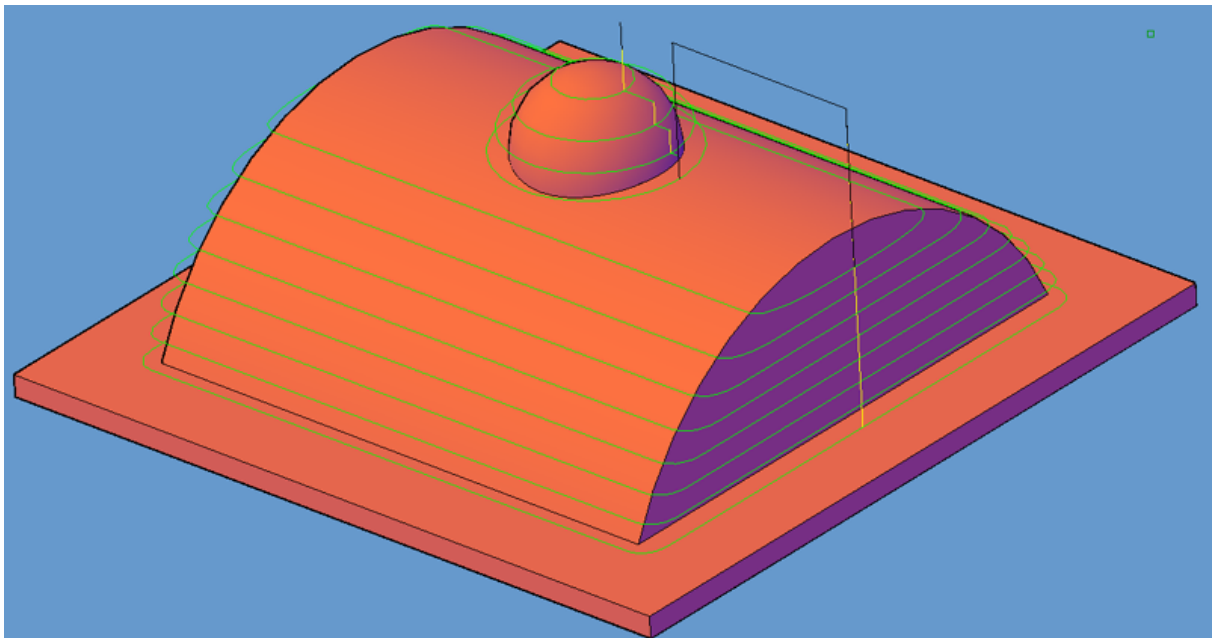
NURBS CCW XY



Nurbs CCW XY is a horizontal slicing cycle. Horizontal slicing will produce a tool path in the XY plane at each Z level, slicing horizontally instead of vertically in either X or Y. This is useful for some roughing and finishing operations where vertical slicing leaves an undesirable finish.

The tool path can be controlled in the Z plane between passes by setting the Z Level Completion option to either Yes or No.

Leads used on this cycle are linear by default with a straight plunge to the cut Z level, however, checking the leads option will allow arc lead-in and lead-out moves to be created for the beginning and end of the tool path. Each slice in between will still have a plunge move in Z.



Shown is a tool path created with the default settings of the NURBS CCW XY cycle. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

```
Enter Surface Tolerance <0.00050000>:
```

```
Enter .005
```

```
Enter Surface Tolerance <0.00050000>: .003
```

You should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

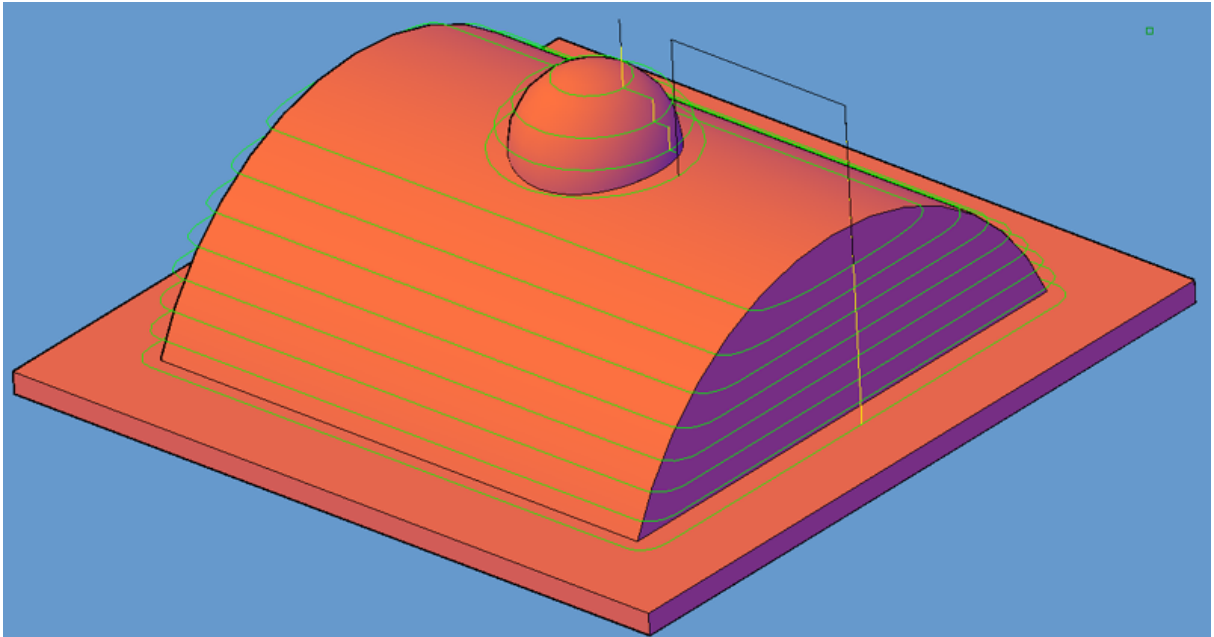
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Horizontal Finishing XY

Select the XNURBS_CUT-CW_XY Cycle. Activate the Cut Button after duplicating the Knowledge Editor entries shown below using MCADTPC.DWG, to achieve similar results. The only parameter changed here is to set the Slice Spacing at .03 to show a more reasonable step over on the tool paths.

Cycle Information	Status Information
<div>Stock Settings</div>	Safety Plane <input type="text" value="*0.25000"/>
<div>Slice Settings</div>	Depth Per Pass <input type="text" value="1.00000"/>
<div>Rough Parameters</div>	Total Cut Depth <input type="text"/>
<div>Containment</div>	Feedrate/Spindle Speed
<div>Recut Settings</div>	Feedrate <input type="text" value="1000.00000"/>
<div>Lead Settings</div>	Spindle Speed <input type="text" value="18000.00000"/>
Material Z Location: <input type="text"/>	Surface FPM <input type="text" value="NONE"/>
	Units Per Revolution <input type="text" value="NONE"/>
	<div>Calculate</div>
<div>Slice Settings</div>	Before Codes <input type="text"/>
Slice Vertical Angle: <input type="text"/>	After Codes <input type="text"/>
Slice Spacing: <input type="text" value="!.tr"/>	Oscillation Amount <input type="text" value="0.00000"/>
<input type="checkbox"/> Constant Scallop	Sort By Rank # <input type="text"/>
<input type="checkbox"/> Lace Slices	
Cut Direction	
<input checked="" type="radio"/> Climb <input type="radio"/> Conventional	
Slice Mode	
<input type="radio"/> ZigZag	
<input type="radio"/> Unidirectional	
<input type="radio"/> Closed Spiral	
<div>OK Cancel</div>	
	<div>NURBS CCW XY</div>
	<div>Reset Cycle Settings to Default</div>

When Prompted for Z Level Completion:



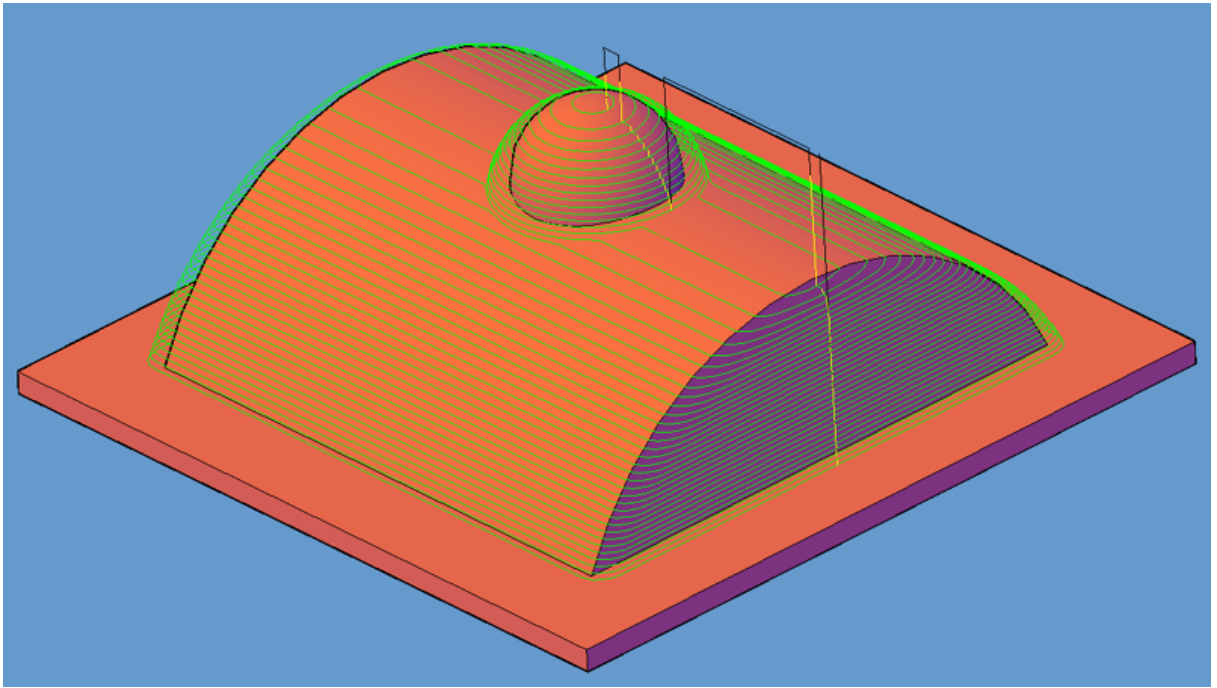
Select **'NO'** to the "Z Level Completion Dialog".

YES is used when a Cavity or Core contains multiple hills and valleys. YES will cause the toolpath calculator to complete all toolpath(s) required at each Z level, retract, and then go down to the next Z level.

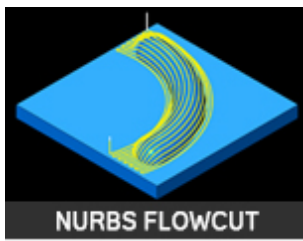
NO continues to cut downward until complete.

Press the **'Cut'** Button.

You should get a tool path like the following:



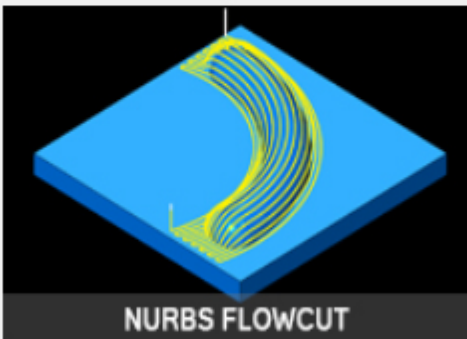
NURBS Flowcut



The Nurbs Flowcut cycle is a contour following slicing cycle. Once the shape has been created, you would typically create two 2D Polylines to be used as flow lines that follow the contour you wish to cut. The Nurbs cutter will then prompt you during the cut to select these flow lines and create a tool path that follows them, on the surface or solid below.

You may use open or closed Polylines for the flow lines and each method is described.

The two Polylines used must be going in the same direction, as the flow cutting tool path will follow these Polylines. This is easily accomplished by using Geoshape and Start Point Edit on both flow lines, which will allow you to make them both become counter-clockwise Polylines starting from the same location.

Cycle Information		Status Information	
<div>Stock Settings</div>		Safety Plane	<input type="text" value="0.25000"/>
<div>Slice Settings</div>		Depth Per Pass	<input type="text" value="1.00000"/>
<div>Rough Parameters</div>		Total Cut Depth	<input type="text"/>
<div>Containment</div>		Feedrate/Spindle Speed	
<div>Recut Settings</div>		Feedrate	<input type="text" value="1000.00000"/>
<div>Lead Settings</div>		Spindle Speed	<input type="text" value="18000.00000"/>
Material Z Location: <input type="text"/>		Surface FPM	<input type="text" value="NONE"/>
		Units Per Revolution	<input type="text" value="NONE"/>
		<div>Calculate</div>	
		Before Codes	<input type="text"/>
		After Codes	<input type="text"/>
		Oscillation Amount	<input type="text" value="0.00000"/>
		Sort By Rank #	<input type="text"/>
			
		<div>Reset Cycle Settings to Default</div>	

Slice Settings

Slice Vertical Angle:

Slice Spacing:

☐ Constant Scallop

☐ Lace Slices

Cut Direction

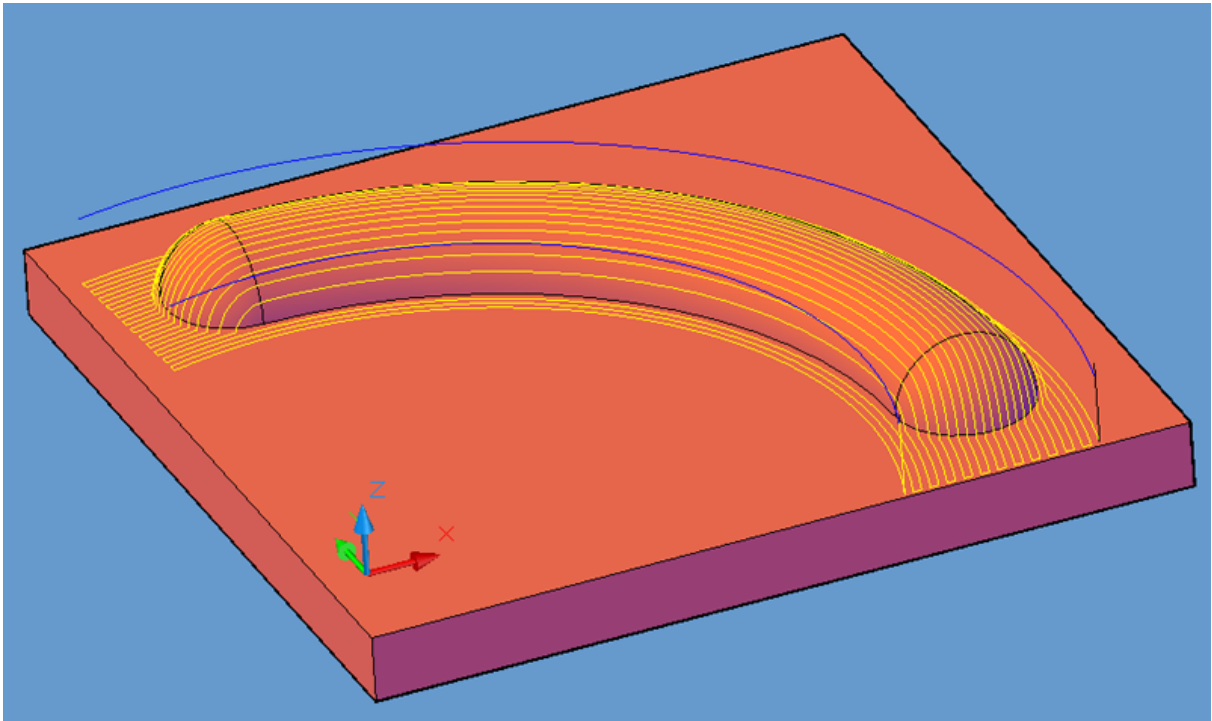
☐ Climb ☒ Conventional

Slice Mode

☒ ZigZag ☐ Unidirectional ☐ Closed Spiral

OK

Cancel



Shown is a tool path created with the default settings of the Nurbs Flowcut cycle on the MCADFLOW.dwg. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

```
Enter Surface Tolerance <0.00050000>:
```

```
Enter .005
```

```
Enter Surface Tolerance <0.00050000>: .003
```

You should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Open Polyline Cross Flow Cutting

Using open polylines that define the flow of your cuts is acceptable as long as both of the polylines are going in the same directions and are above the top of the part. You may use Geoshape and Start Point Edit if you need to be sure of the direction of you flow lines.

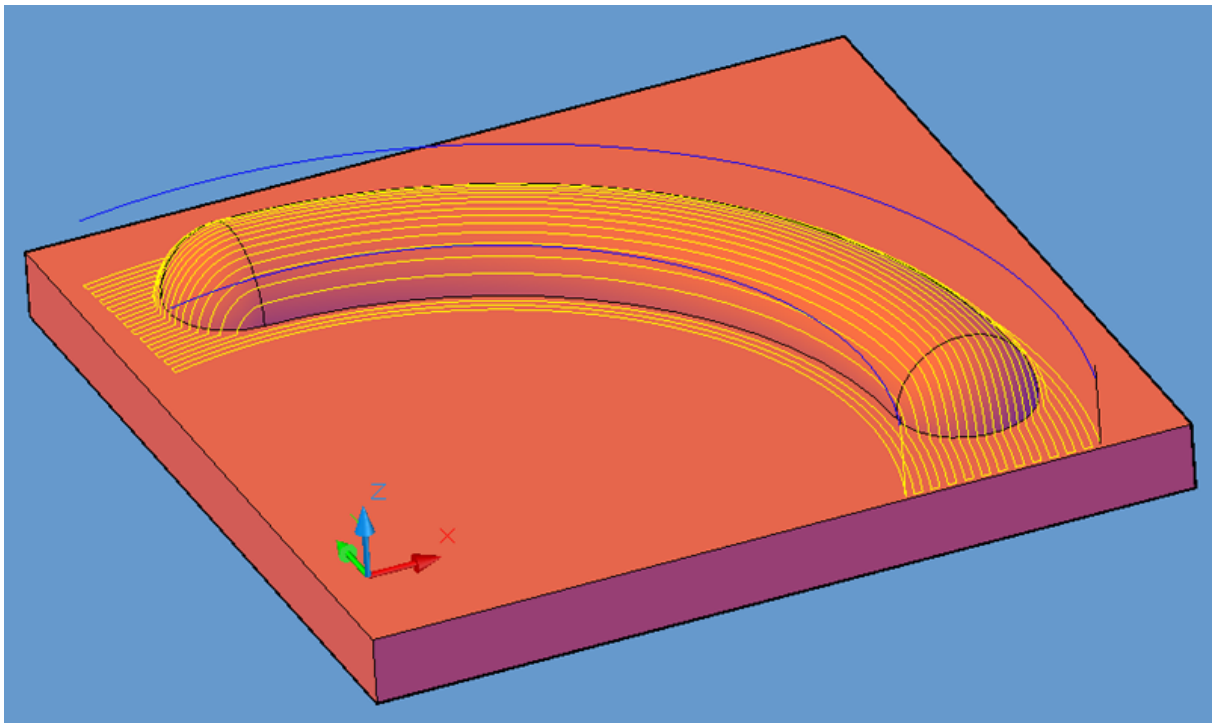
During cut you will be prompted not only to select the surface or solid to cut, but for your flow lines as well.

Using the MCADFLOW.dwg file, select the XNURBS_CUT-FLOWCUT Cycle.
Select Slice Settings to be sure the Slice Vertical Angle is set to 0.0, and the Slice Mode is Zigzag.

The screenshot displays the Router-CIM software interface. On the left, the 'Cycle Information' panel contains a list of settings: Stock Settings, Slice Settings, Rough Parameters, Containment, Recut Settings, and Lead Settings. Below this list is a 'Material Z Location' field. A 'Slice Settings' dialog box is open in the foreground, showing the following options: 'Slice Vertical Angle' set to 0, 'Slice Spacing' set to '!tr*', 'Constant Scallop' and 'Lace Slices' are unchecked, 'Cut Direction' is set to 'Conventional' (radio button selected), and 'Slice Mode' is set to 'ZigZag' (radio button selected). The 'OK' and 'Cancel' buttons are at the bottom of the dialog. On the right, the 'Status Information' panel shows various parameters: 'Safety Plane' at *0.25000, 'Depth Per Pass' at 1.00000, 'Total Cut Depth' is empty, 'Feedrate/Spindle Speed' section includes 'Feedrate' at 1000.00000, 'Spindle Speed' at 18000.00000, 'Surface FPM' at NONE, and 'Units Per Revolution' at NONE. A 'Calculate' button is present. Below this, 'Before Codes', 'After Codes', 'Oscillation Amount' (0.00000), and 'Sort By Rank #' are shown. At the bottom right, there is a 3D visualization of a blue rectangular block with a yellow curved tool path, labeled 'NURBS FLOWCUT'. A 'Reset Cycle Settings to Default' button is located below the visualization.

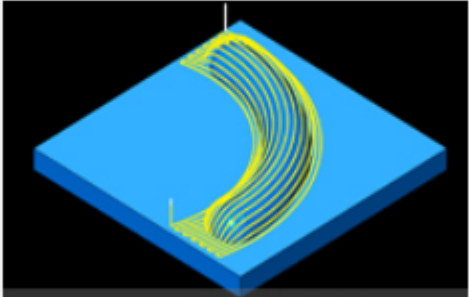
Press the '**Cut**' Button.

The output of these settings should produce a tool path like this:



Slice Angle of 0.

Erase the previous tool path, then change the Slice Settings Vertical Angle to 90.0.

Cycle Information		Status Information	
Stock Settings		Safety Plane	*0.25000
Slice Settings		Depth Per Pass	1.00000
Rough Parameters		Total Cut Depth	
Containment		Feedrate/Spindle Speed	
Recut Settings		Feedrate	1000.00000
Lead Settings		Spindle Speed	18000.00000
Material Z Location:		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		NURBS FLOWCUT	
		Reset Cycle Settings to Default	

Slice Settings

Slice Vertical Angle: 90

Slice Spacing: !*tr*

☐ Constant Scallop

☐ Lace Slices

Cut Direction

☐ Climb

☒ Conventional

Slice Mode

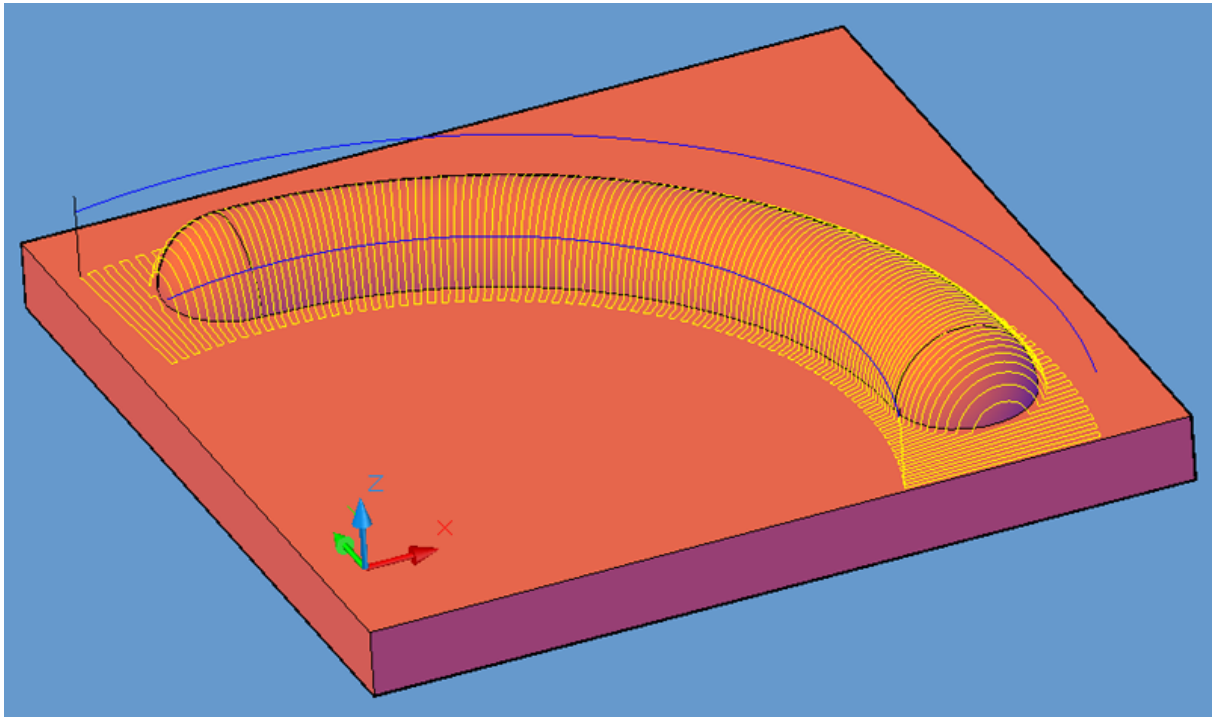
☒ ZigZag

☐ Unidirectional

☐ Closed Spiral

OK

Cancel



Slice Angle of 90.

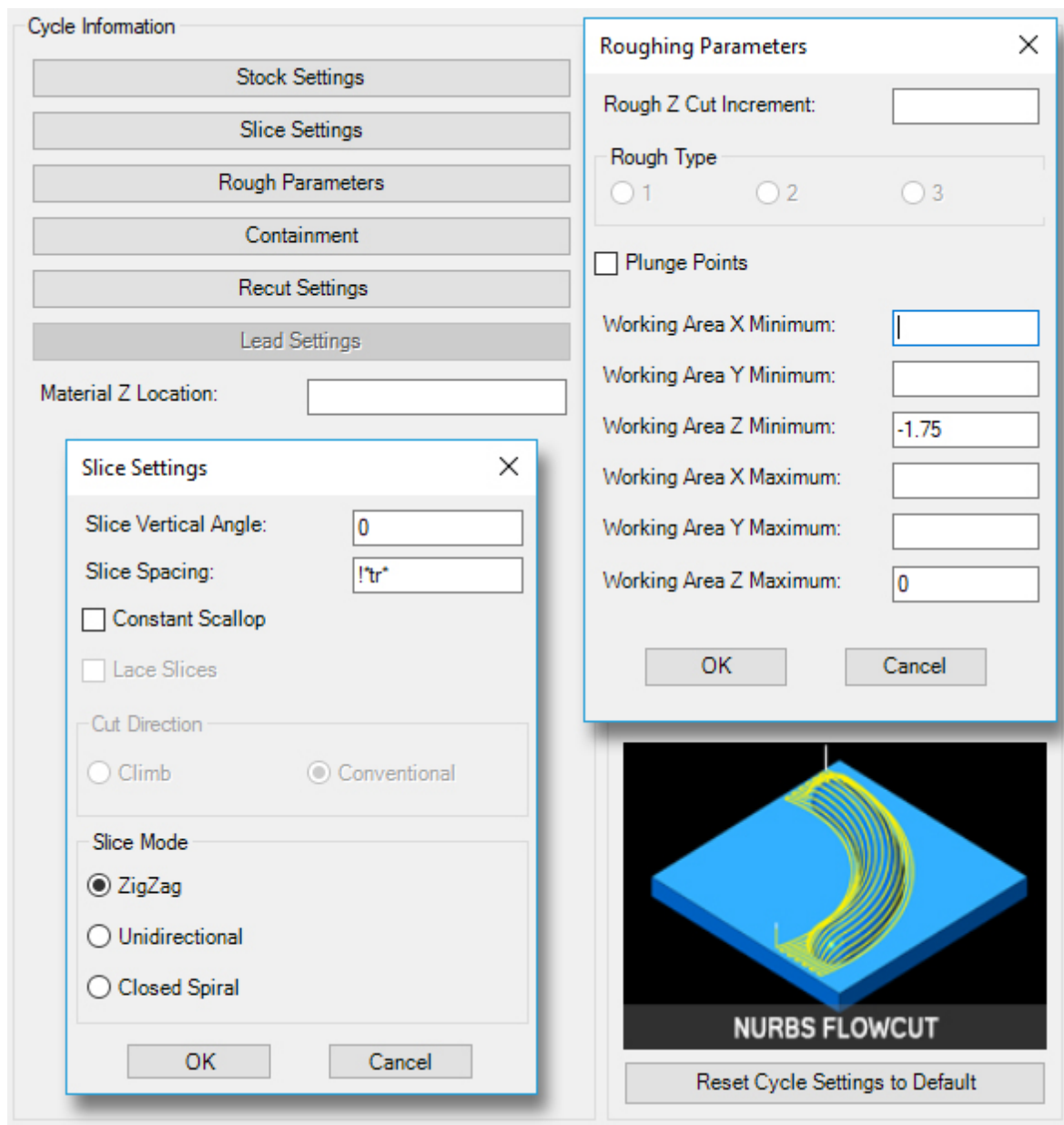
Closed Polyline Step Over Flow Cutting

Closed Polyline Flow Cutting is provided to satisfy cutting situations best handled by a continuous circulating toolpath, that is guided within an area defined by two closed polylines. Creating the two polylines is typically this is accomplished by creating a polyline from the edge of the solid and then offsetting it the appropriate amount for the tool. These closed polylines must be going in the same direction and the Start Points of both polylines must be aligned. These conditions are easily met by using Geoshape and Start Point Edit on the two polylines planned for use with this cycle. Geoshape will ensure the polylines are going in the counterclockwise directions and Start Point Edit will allow you to align the two start points.

The Flow Cutting cycle will follow the direction of the two polylines, so you can use the command NCREVDIR to reverse the direction of the two polylines if desired.

Since there is nothing to contain the tool path when it meets the bottom of the shape, we will use the NURBS Working Area settings to define the top and bottom of the shape so that the tool path does not go below the bottom of the part.

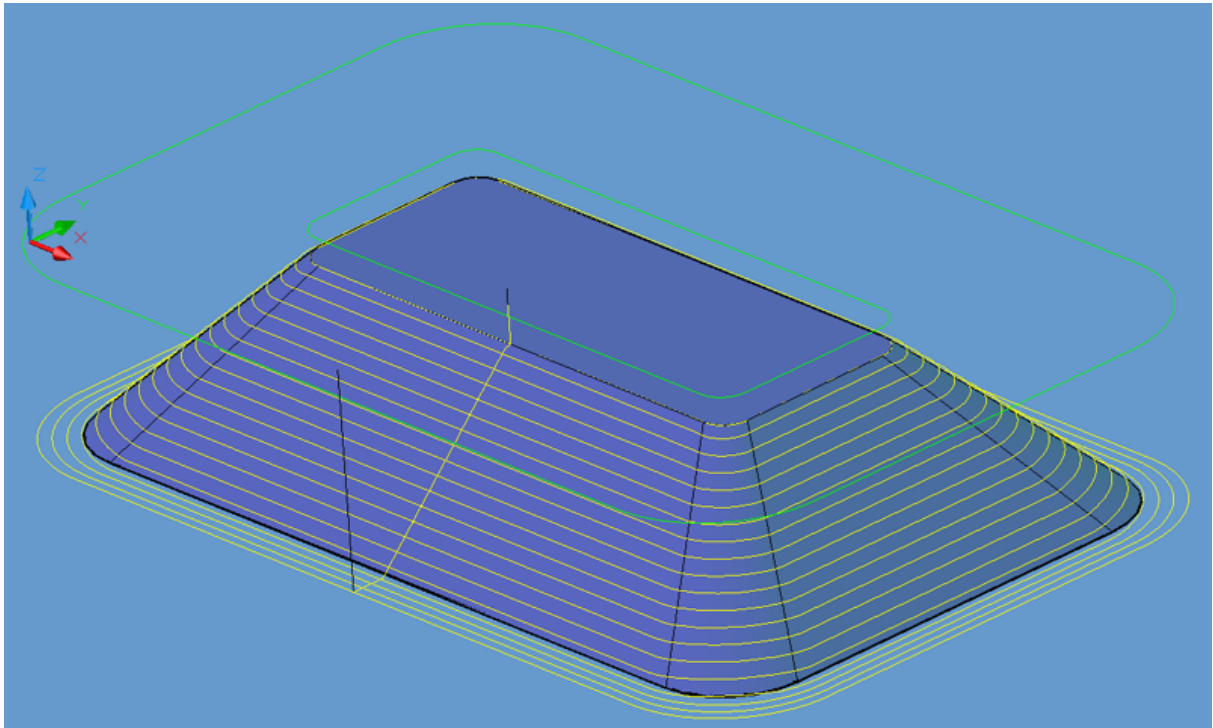
Open MCADCLFLOW.dwg and start Router-CIM. Change your knowledge settings to match the ones below:



The Roughing Parameters window will allow you to set the Working Area minimum and maximum so that no tool paths cross beyond these areas. Set the Z Maximum to 0.0 which is the top of the part and the Z Minimum to -1.75 which is the bottom of the part and then no tool paths will fall below the part.

Setting the Slice Mode to Zigzag will allow the tool path to make a step down between each pass without lifting up to the Safety Plane between passes.

Change your settings to match the ones here and then make a cut, selecting the solid as the shape to cut and the two green polylines as the Flow Lines. If prompted to redefine the surfaces, answer Yes. Your results should look like the one below:



This tool path will step down in Z for each pass without lifting the tool up to the Safety Plane, and continue cutting the solid until it gets to the bottom of the shape at -1.75. Then, since it is still cutting until it meets the larger of the two closed flow lines, it will make passes in XY. Making a smaller outside flow line, that matches the shape would avoid the extra passes at the bottom.

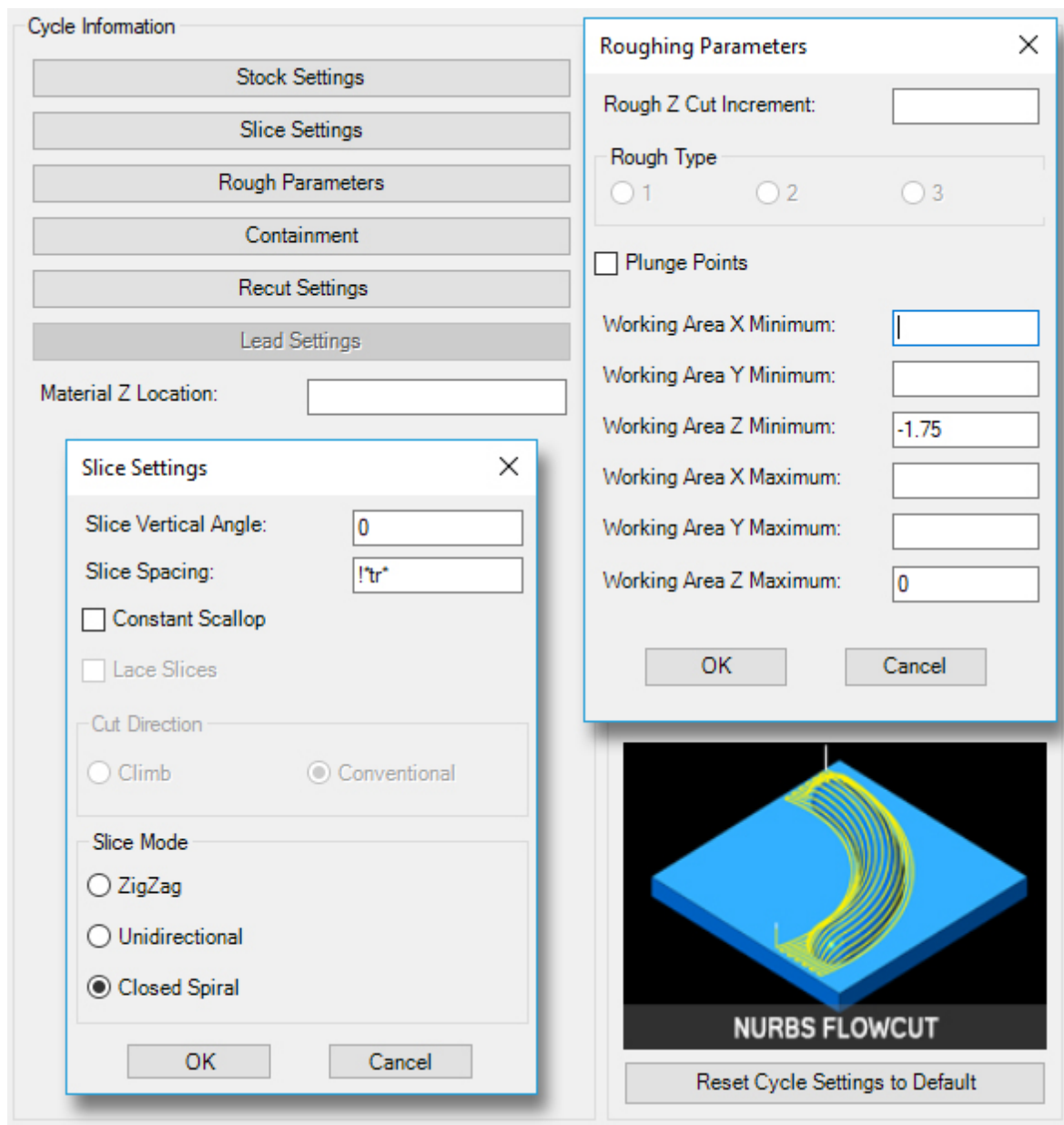
Closed Polyline Spiral Flow Cutting

Closed Polyline Flow Cutting is provided to satisfy cutting situations best handled by a continuous spiraling toolpath, that is guided within an area defined by two closed polylines. Creating the two polylines is typically this is accomplished by creating a polyline from the edge of the solid and then offsetting it the appropriate amount for the tool. These closed polylines must be going in the same direction and the Start Points of both polylines must be aligned. These conditions are easily met by using Geoshape and Start Point Edit on the two polylines planned for use with this cycle. Geoshape will ensure the polylines are going in the counterclockwise directions and Start Point Edit will allow you to align the two start points.

The Flow Cutting cycle will follow the direction of the two polylines, so you can use the command NCREVDIR to reverse the direction of the two polylines if desired.

Since there is nothing to contain the tool path when it meets the bottom of the shape, we will use the NURBS Working Area settings to define the top and bottom of the shape so that the tool path does not go below the bottom of the part.

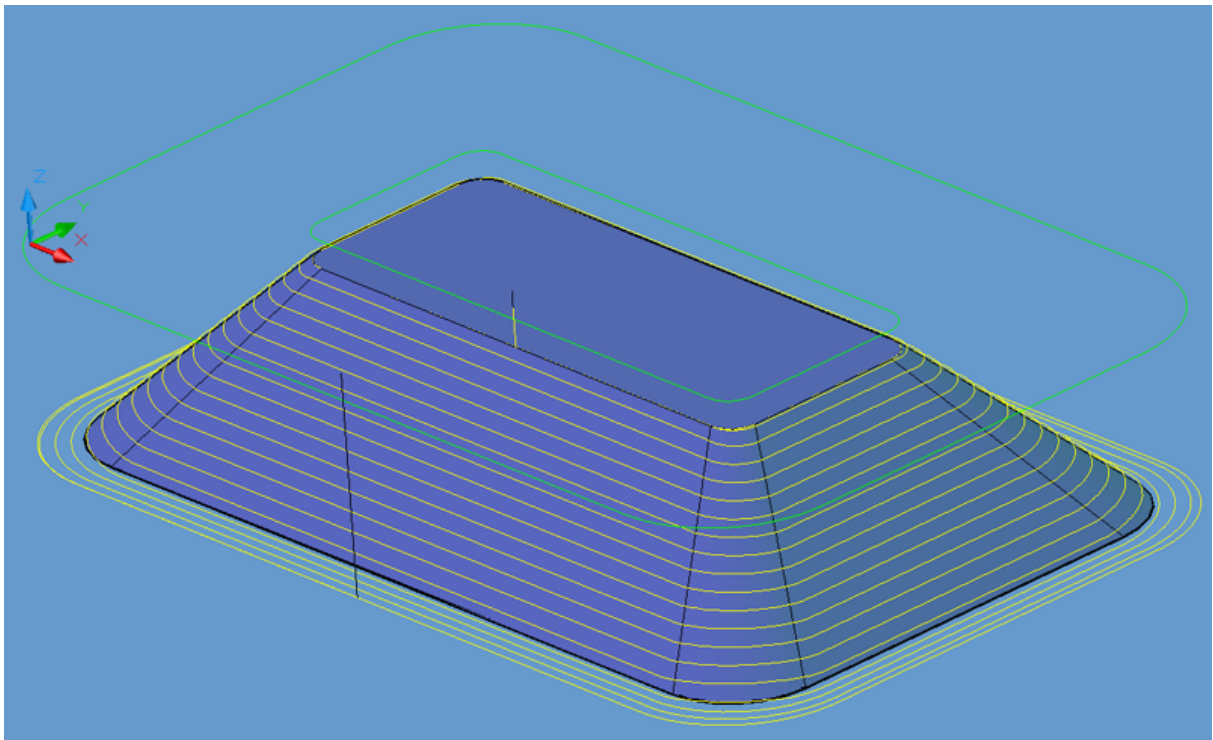
Open MCADCLFLOW.dwg and start Router-CIM. Change your knowledge settings to match the ones below:



The Roughing Parameters window will allow you to set the Working Area minimum and maximum so that no tool paths cross beyond these areas. Set the Z Maximum to 0.0 which is the top of the part and the Z Minimum to -1.75 which is the bottom of the part and then no tool paths will fall below the part.

Setting the Slice Mode to Closed Spiral will allow the tool path to make a constant spiral pass without lifting up to the Safety Plane until it reaches the end of the path.

Change your settings to match the ones here and then make a cut, selecting the solid as the shape to cut and the two green polylines as the Flow Lines. If prompted to redefine the surfaces, answer Yes. Your results should look like the one below:



This tool path starts at the top of the part, on the boundary of the inner flow line, and makes one tool path constantly stepping down in Z while circling the part until it reaches the minimum Z depth set in the Roughing Parameters and then continues at that depth until it reaches the outside flow line.

Closed Polyline Cross Flow Cutting

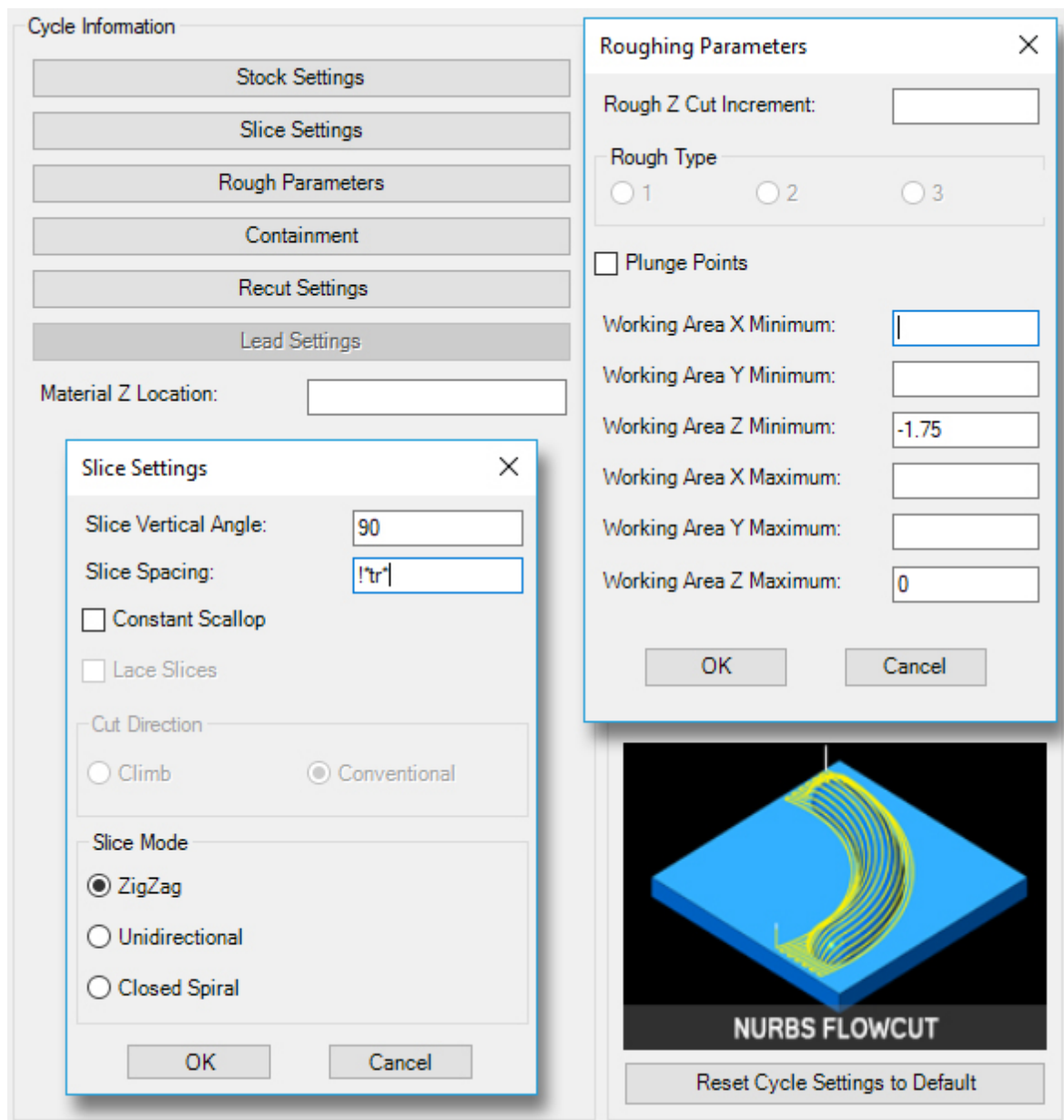
Closed Polyline Flow Cutting is provided to satisfy cutting situations best handled by a toolpath cutting at 90° to the flow lines provided on the part. Creating the two polylines is typically this is accomplished by creating a polyline from the edge of the solid and then offsetting it the appropriate amount for the tool.

These closed polylines must be going in the same direction and the Start Points of both polylines must be aligned. These conditions are easily met by using Geoshape and Start Point Edit on the two polylines planned for use with this cycle. Geoshape will ensure the polylines are going in the counterclockwise directions and Start Point Edit will allow you to align the two start points.

The Flow Cutting cycle will cut perpendicular to the flow lines, but cutting in the direction of the two polylines, so you can use the command NCREVDIR to reverse the direction of the two polylines if desired.

Since there is nothing to contain the tool path when it meets the bottom of the shape, we will use the NURBS Working Area settings to define the top and bottom of the shape so that the tool path does not go below the bottom of the part.

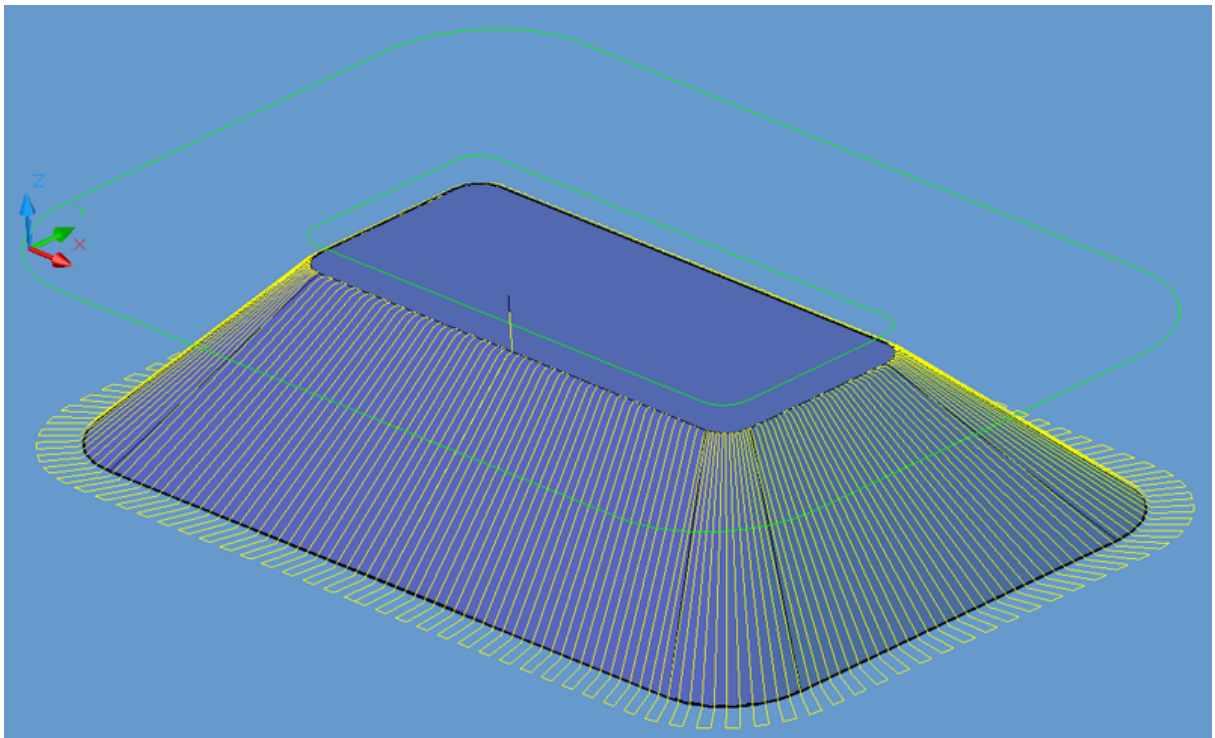
Open MCADCLFLOW.dwg and start Router-CIM. Change your knowledge settings to match the ones below:



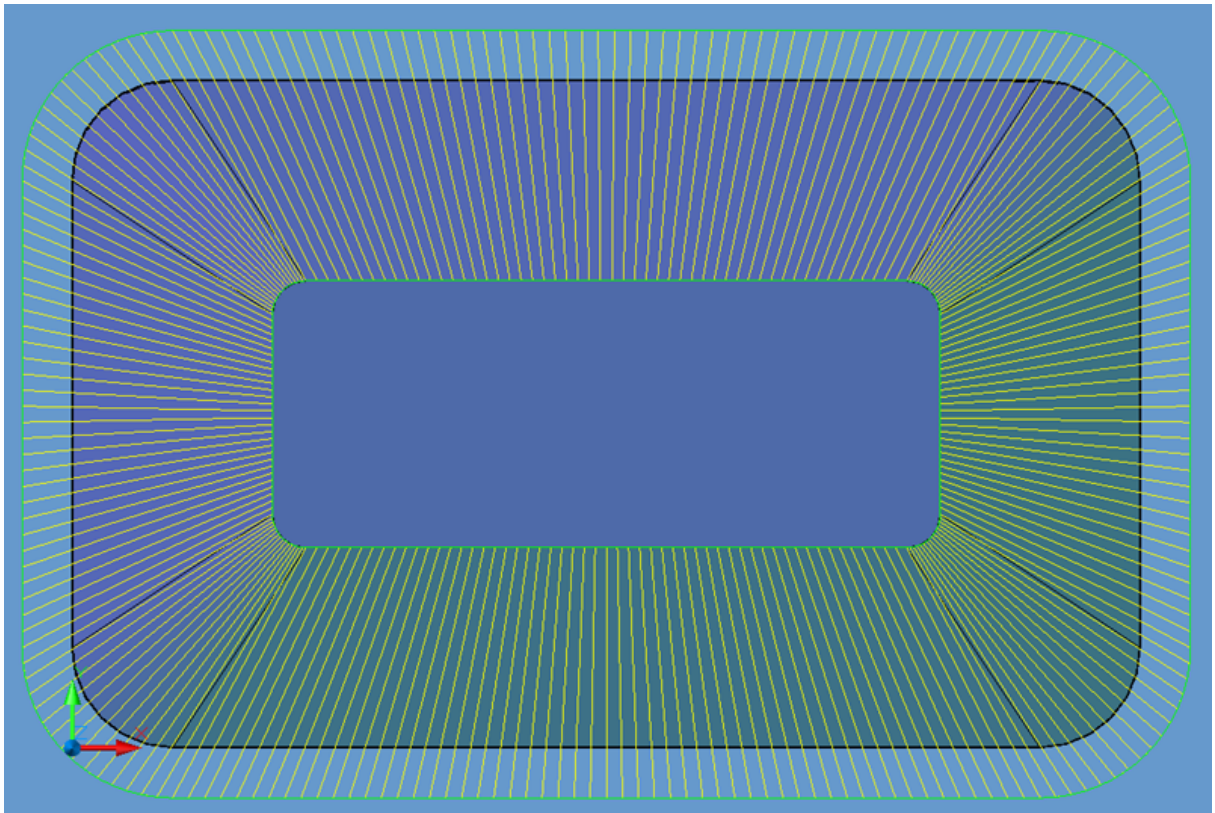
The Roughing Parameters window will allow you to set the Working Area minimum and maximum so that no tool paths cross beyond these areas. Set the Z Maximum to 0.0 which is the top of the part and the Z Minimum to -1.75 which is the bottom of the part and then no tool paths will fall below the part.

Setting the Slice Mode to Zigzag and the Slice Vertical Angle will allow the tool path to make a evenly spaced tool paths without lifting up to the Safety Plane until it reaches the end of the path.

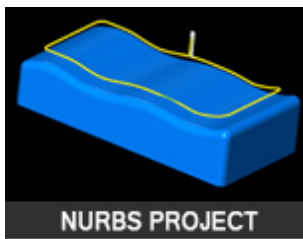
Change your settings to match the ones here and then make a cut, selecting the solid as the shape to cut and the two green polylines as the Flow Lines. If prompted to redefine the surfaces, answer Yes. Your results should look like the one below:



The result of this tool path is evenly spaced tool paths following the contours of the flow lines.

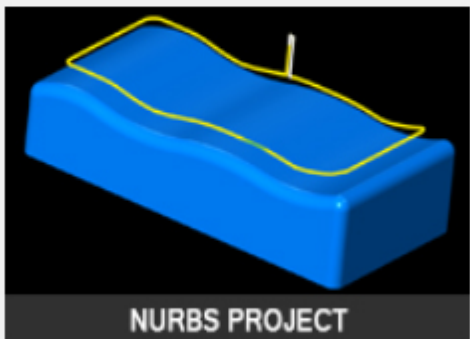


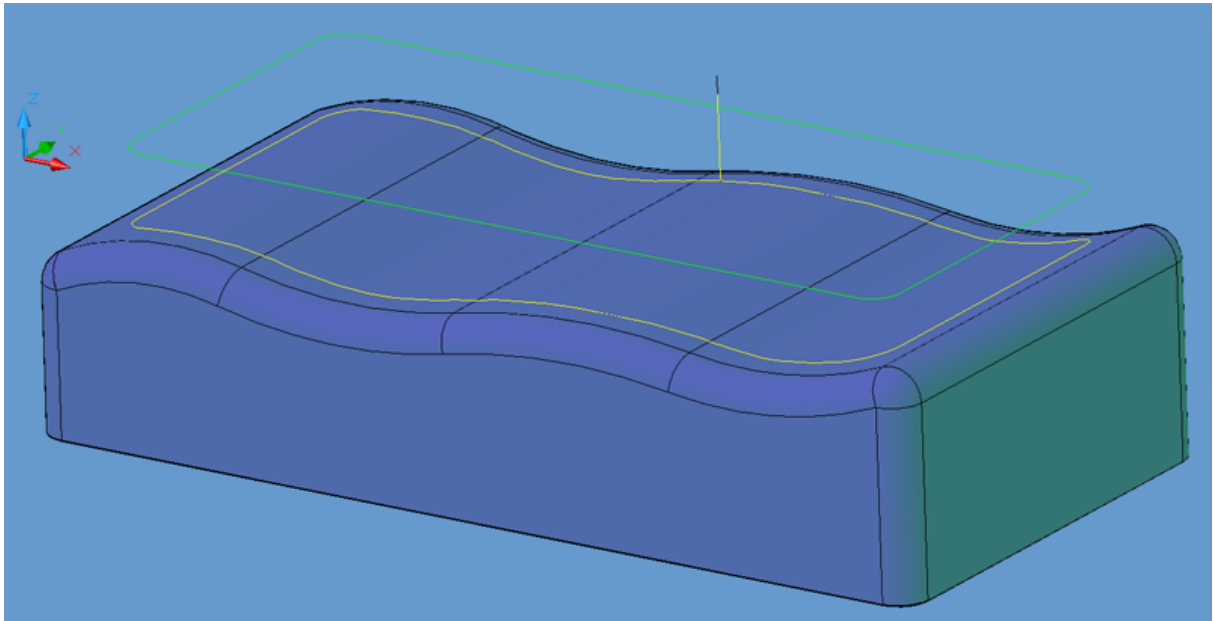
NURBS Project



Project Cutting will create a tool path that is projected from a polyline(s) drawn above the part, down onto the surfaces or solid representing the part. Multiple polylines can be projected at one time. The resulting tool paths are vertical tool paths following the contour of the surface or solid.

The resulting tool path is on the top of the surface, but if you wish to move the tool path into the material, you can set a Stock Allowance of a Negative value to move the tool path down into the material.

Cycle Information	Status Information
<div>Stock Settings</div>	Safety Plane <input type="text" value="*0.25000"/>
<div>Slice Settings</div>	Depth Per Pass <input type="text" value="1.00000"/>
<div>Rough Parameters</div>	Total Cut Depth <input type="text"/>
<div>Containment</div>	Feedrate/Spindle Speed
<div>Recut Settings</div>	Feedrate <input type="text" value="350.00000"/>
<div>Lead Settings</div>	Spindle Speed <input type="text" value="18000.00000"/>
Material Z Location: <input type="text"/>	Surface FPM <input type="text" value="NONE"/>
	Units Per Revolution <input type="text" value="NONE"/>
	<div>Calculate</div>
	Before Codes <input type="text"/>
	After Codes <input type="text"/>
	Oscillation Amount <input type="text" value="0.00000"/>
	Sort By Rank # <input type="text"/>
	<div> NURBS PROJECT</div>
	<div>Reset Cycle Settings to Default</div>



Shown is a tool path created with the default settings of the Nurbs Project cycle. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

```
Enter Surface Tolerance <0.00050000>:
```

```
Enter .005
```

```
Enter Surface Tolerance <0.00050000>: .003
```

You should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

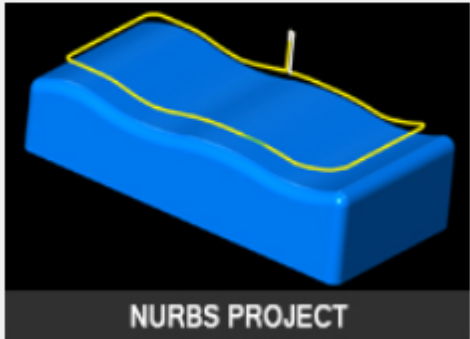
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Polyline Project Cutting

Using Polyline Project Cutting, we can take several flat shapes and project them down onto an irregular surface or solid.

Open the drawing NurbsProject_1.dwg and start Router-CIM.

Using a .25 Ball End Mill and the default cycle settings, press 'Cut'.

Cycle Information	Status Information
<input type="button" value="Stock Settings"/>	Safety Plane <input type="text" value="0.25000"/>
<input type="button" value="Slice Settings"/>	Depth Per Pass <input type="text" value="1.00000"/>
<input type="button" value="Rough Parameters"/>	Total Cut Depth <input type="text"/>
<input type="button" value="Containment"/>	Feedrate/Spindle Speed
<input type="button" value="Recut Settings"/>	Feedrate <input type="text" value="350.00000"/>
<input type="button" value="Lead Settings"/>	Spindle Speed <input type="text" value="18000.00000"/>
Material Z Location: <input type="text"/>	Surface FPM <input type="text" value="NONE"/>
	Units Per Revolution <input type="text" value="NONE"/>
	<input type="button" value="Calculate"/>
	Before Codes <input type="text"/>
	After Codes <input type="text"/>
	Oscillation Amount <input type="text" value="0.00000"/>
	Sort By Rank # <input type="text"/>
	
	<input type="button" value="Reset Cycle Settings to Default"/>

Select the blue solid when prompted to select the part.

When prompted to select the Surface Tolerance, enter .005.

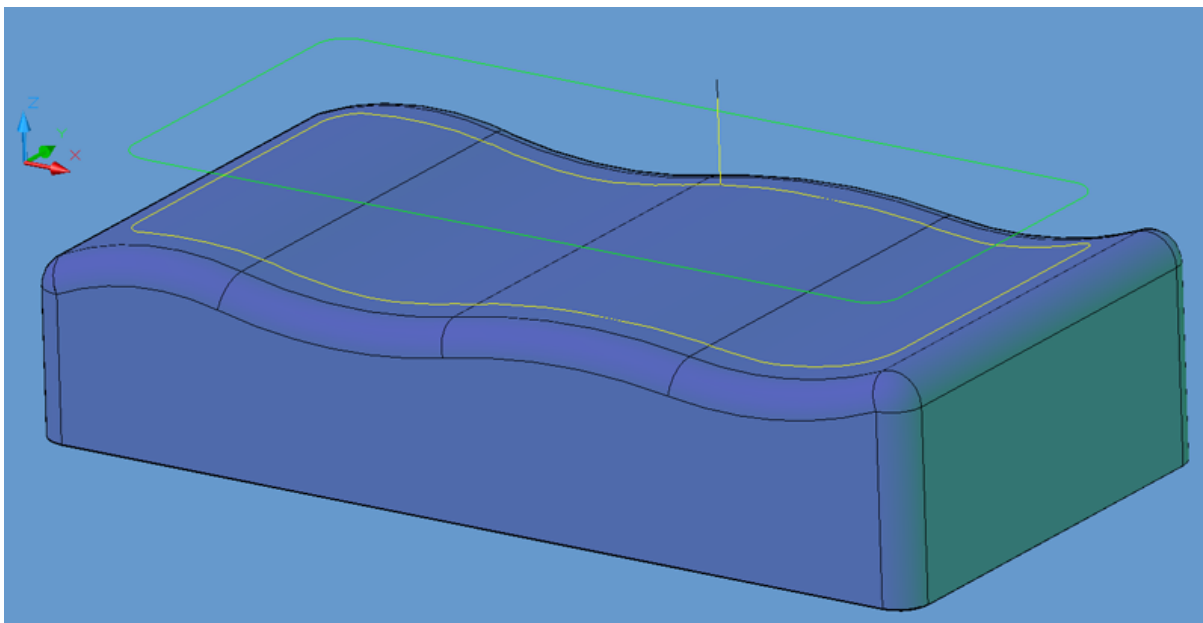
Enter Surface Tolerance <0.00050000>: .005

Next you will be prompted to select the Projection Polylines.

Select Projection Polylines/Cut Blocks:

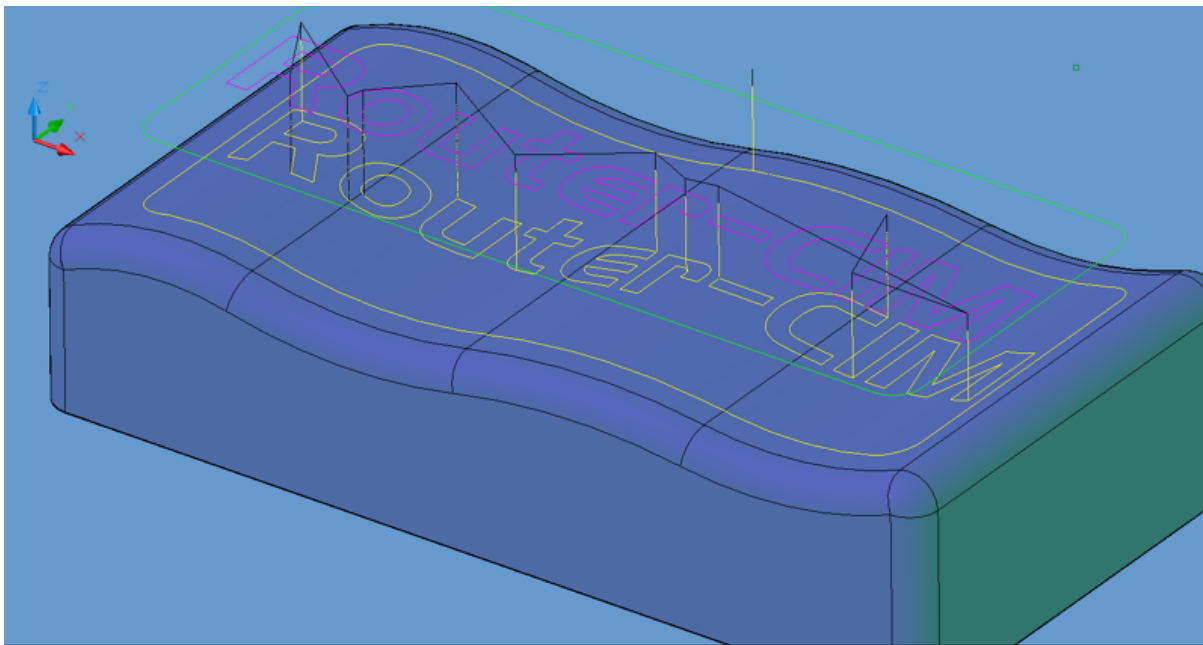
Select the green polyline above the part.

You should then see a tool path similar to the one shown here:



If you wish to make more cuts, Thaw layer ProjectText1, and then press **'Cut'** again, reselecting the solid and selecting the magenta text as the Projection Polylines.

You should end up with a tool path similar to this:



Specialty Cycles

There are some specialty cycles built into Router-CIM to accomplish specific tasks. These cycles are not useful on all machines. For instance there is a 4 Axis Saw cycle, and if your machine tool is not equipped with this type of spindle, then the cycle will not produce any suitable results.

Each of the specialty cycles are described in some detail to give an idea as to how the cycle works and when it is useful.

4th Axis Machining

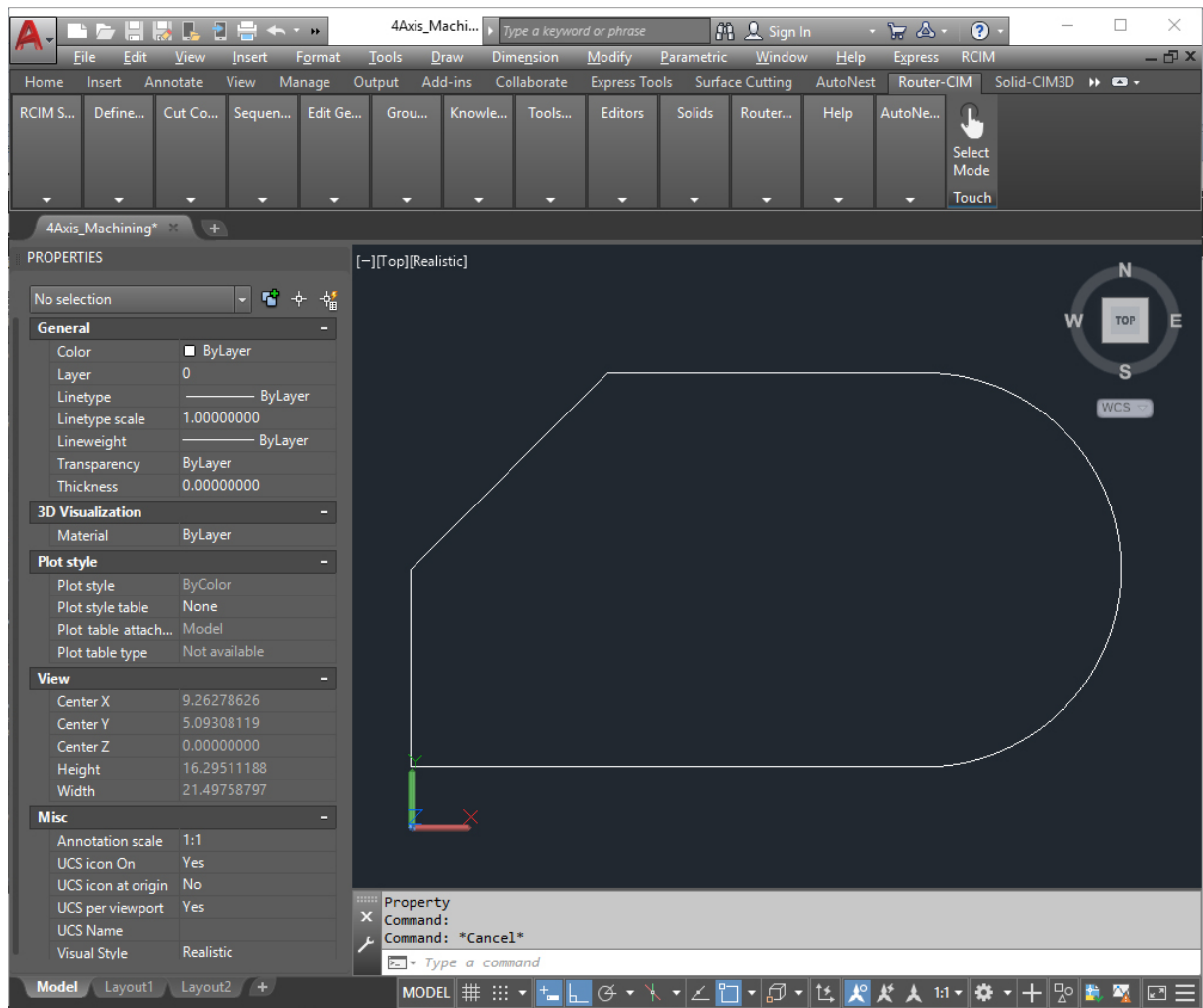
There are some specialty cycles built into Router-CIM to accomplish specific tasks. These cycles are not useful on all machines. For instance there is a 4 Axis Saw cycle, and if your machine tool is not equipped with this type of spindle, then the cycle will not produce any suitable results.

Parts to be cut with a fourth axis may be drawn as you normally would draw a 2-D part unless you are using a standard cut cycle, heli-inside, pocketing, drill motions, etc. In order to use a standard cut cycle, you will need to draw on a surface other than the top by changing the User Coordinate System (UCS) to draw on each face and add thickness to the geometry that is drawn to insure you are on the correct face of the part.

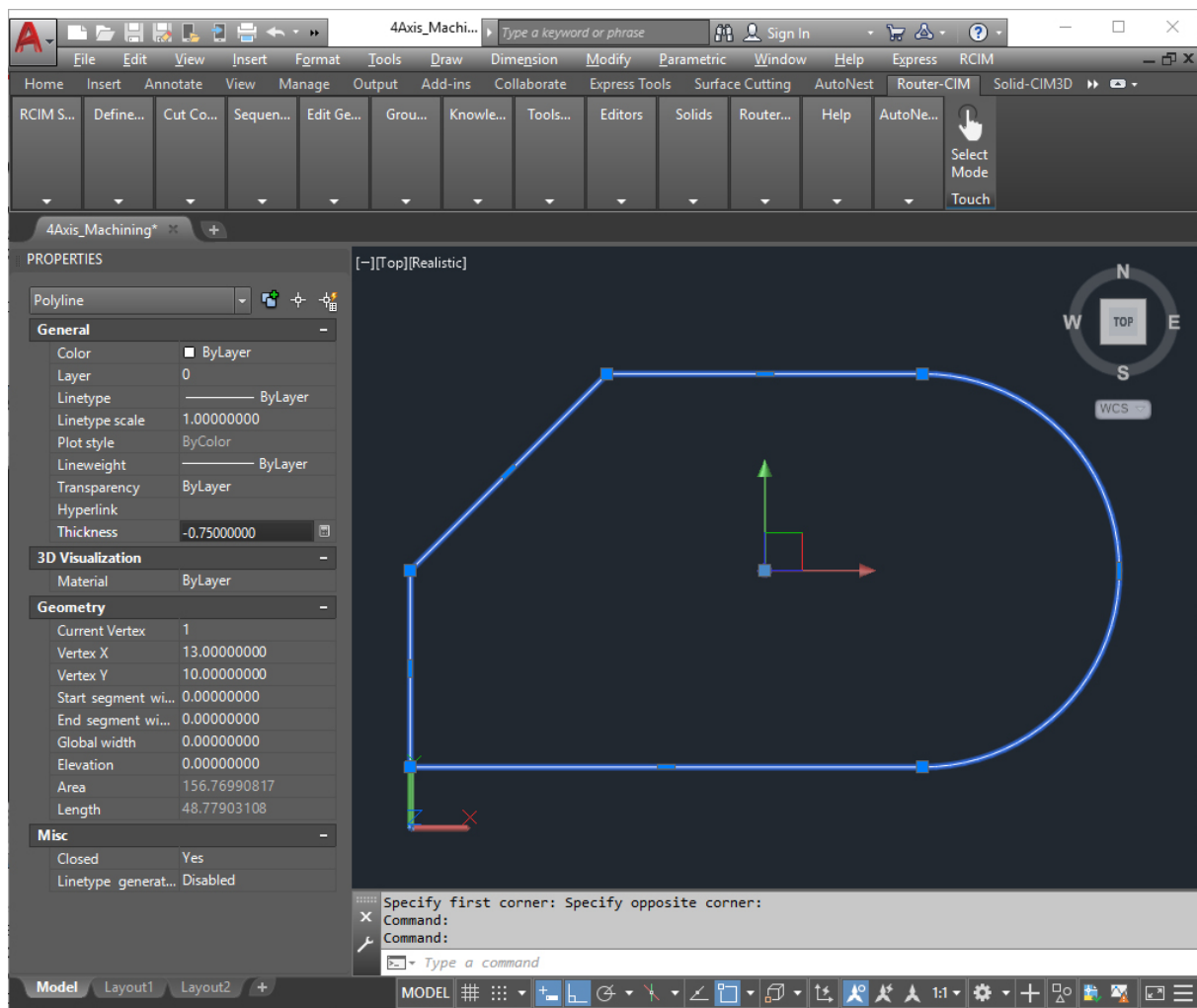
There are 2 main methods when drawing on a horizontal face, Planar (0, 90, 180, 270) and Non-Planar (Any degree).

How to draw on a face/side of a part (Planar):

1. Draw the part to be cut as you normally would.
2. Add thickness to the part as described in the process below.



- A. Choose the properties manager icon from your "Standard Toolbar" or press Ctrl and 1 at the same time.
 - 1) Move or dock this window to the side of your drawing area so you can see your geometry.
 - 2) Notice that the field at the top of this window says "No Selection"
- B. Select the whole part by clicking on each segment or making a window around it.
 - 1) Notice that the field at the top of this window now tells you what and how many segments have been selected.



Always Enter Thickness as Negative Value

C. Find the "Thickness" field in this window and change the value to the actual thickness of your part or material. The value should always be a negative. Press the ENTER key to activate the change.

1) To remove the grips (blue squares) from the geometry, move the cursor into the drawing area and press the ESCAPE key.

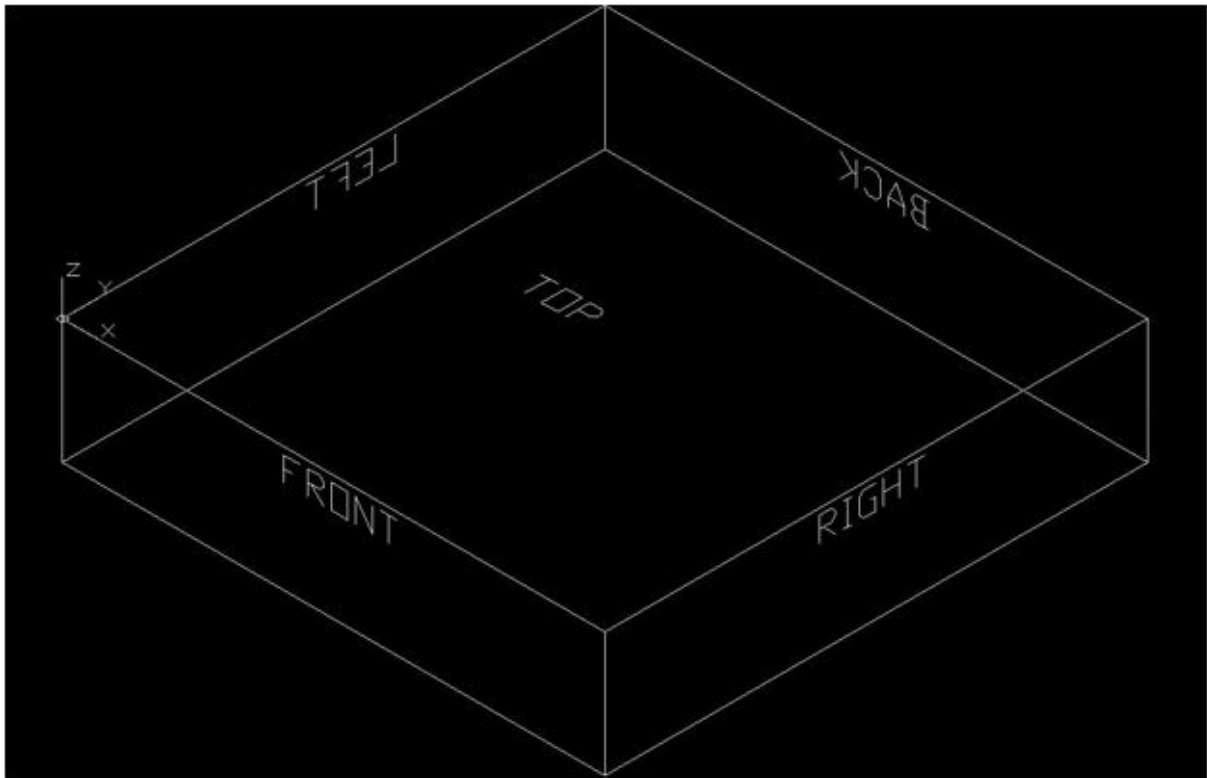
D Change the view to a SE isometric so that you can see the thickness of the part.

3. Change the UCS to match your working surface/plane. All horizontal cuts to be made on the side of the part must be drawn in the proper UCS.

A. Start Router-CIM

B. Decide which side of the part you would like to draw on.

1) If you were to look at a rectangle in a top view, you would see four sides. A Left Side (LS), A Right Side (RS), A Back Side (BS) would be towards the top, and the Front Side (FS) would be towards the bottom.



C. Type the abbreviation of the side you wish to draw a cut on at the command line. RS, LS, FS, or BS.

- D. Select the origin point. The origin point is going to become the base of the UCS icon
- 1) If you were to face the side you want to cut on, choose the bottom left corner for the origin. The Origin is Always Lower Left!
 - 2) The Endpoint Snap is automatically turned on. When you move the cursor close to the bottom left corner the Endpoint Snap will light up. At this point click to set the UCS. Notice the UCS icon has changed so that the X and Y directions are towards the center of this surface and the Z is pointing away from this surface.

4. Draw the geometry on this surface as you normally would in a top view.

5. Repeat steps 4 and 5 as needed for each side.

How to draw on a face/side of a part (Non-Planar):

1. Go to the pull down menus at the top of the screen and choose "Tools", "New UCS", "3point" or type UCS in the command line. The default is 3point.

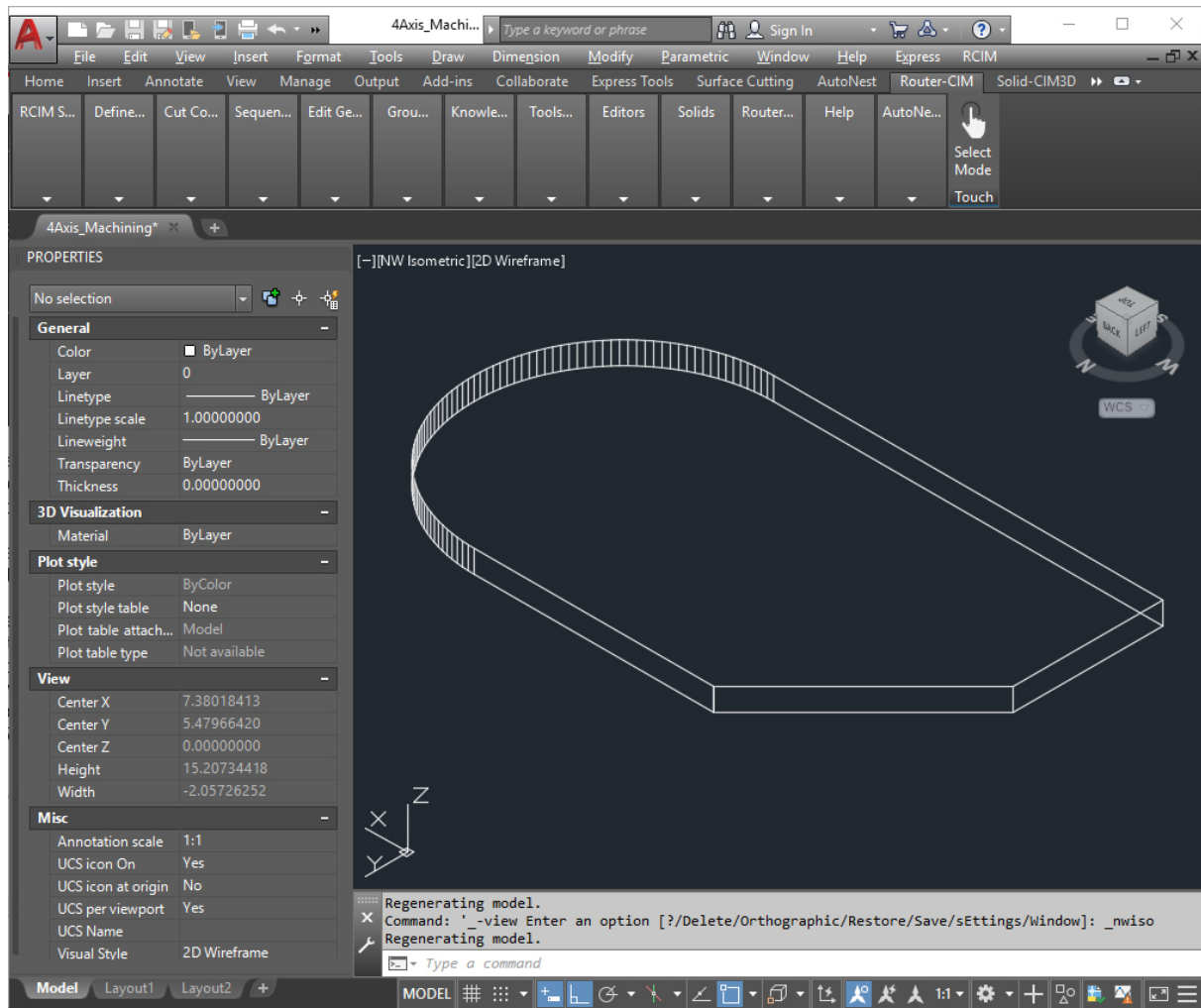
For more information on changing the UCS in AutoCAD, please refer to the AutoCAD Help Manual.

2. Select the origin point. The origin point will become the base of the UCS icon.

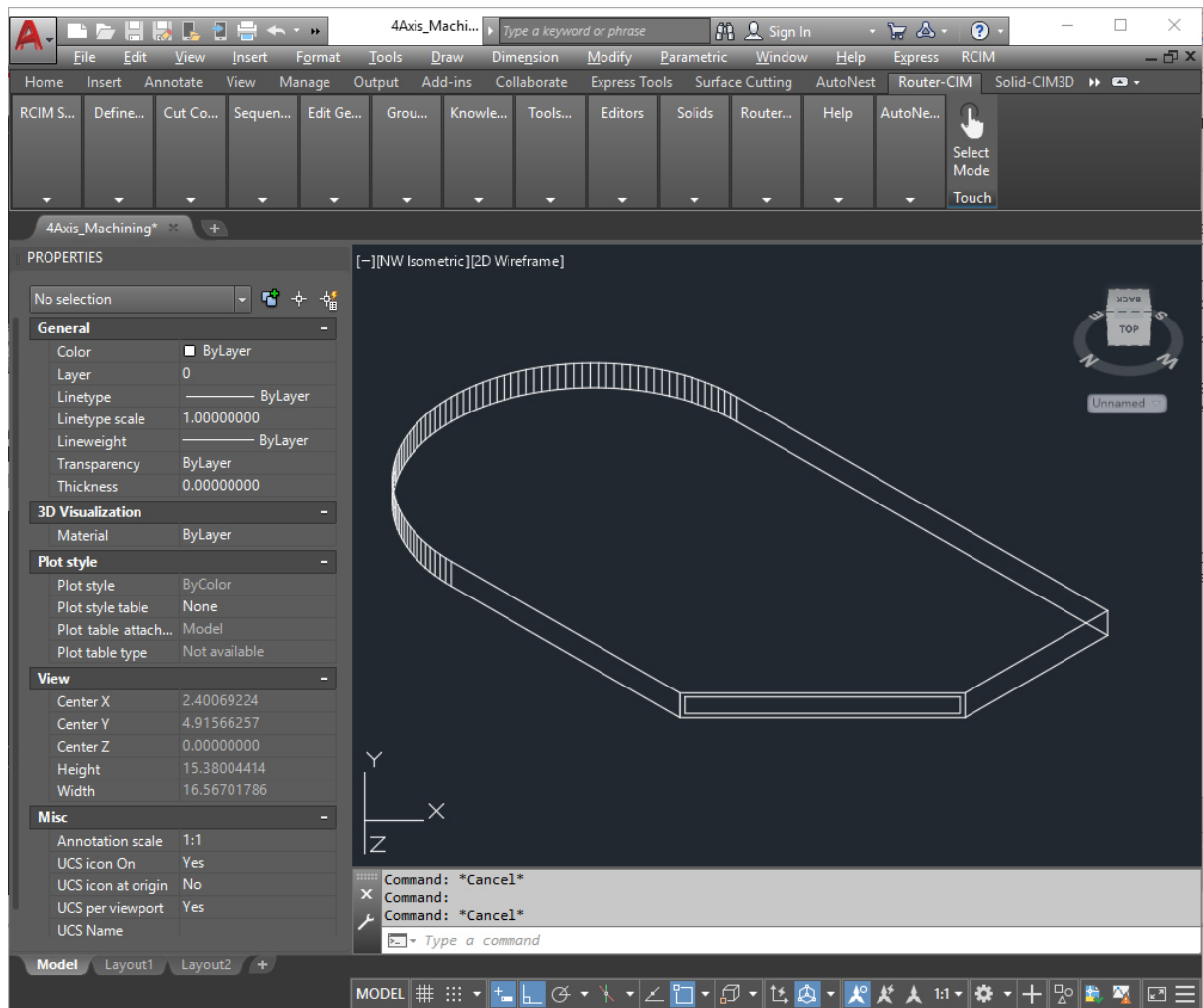
A. If you were to face this surface choose the bottom left corner as the origin. Choose the origin with an endpoint snap.

3. Choose a point on the positive portion of the X axis.

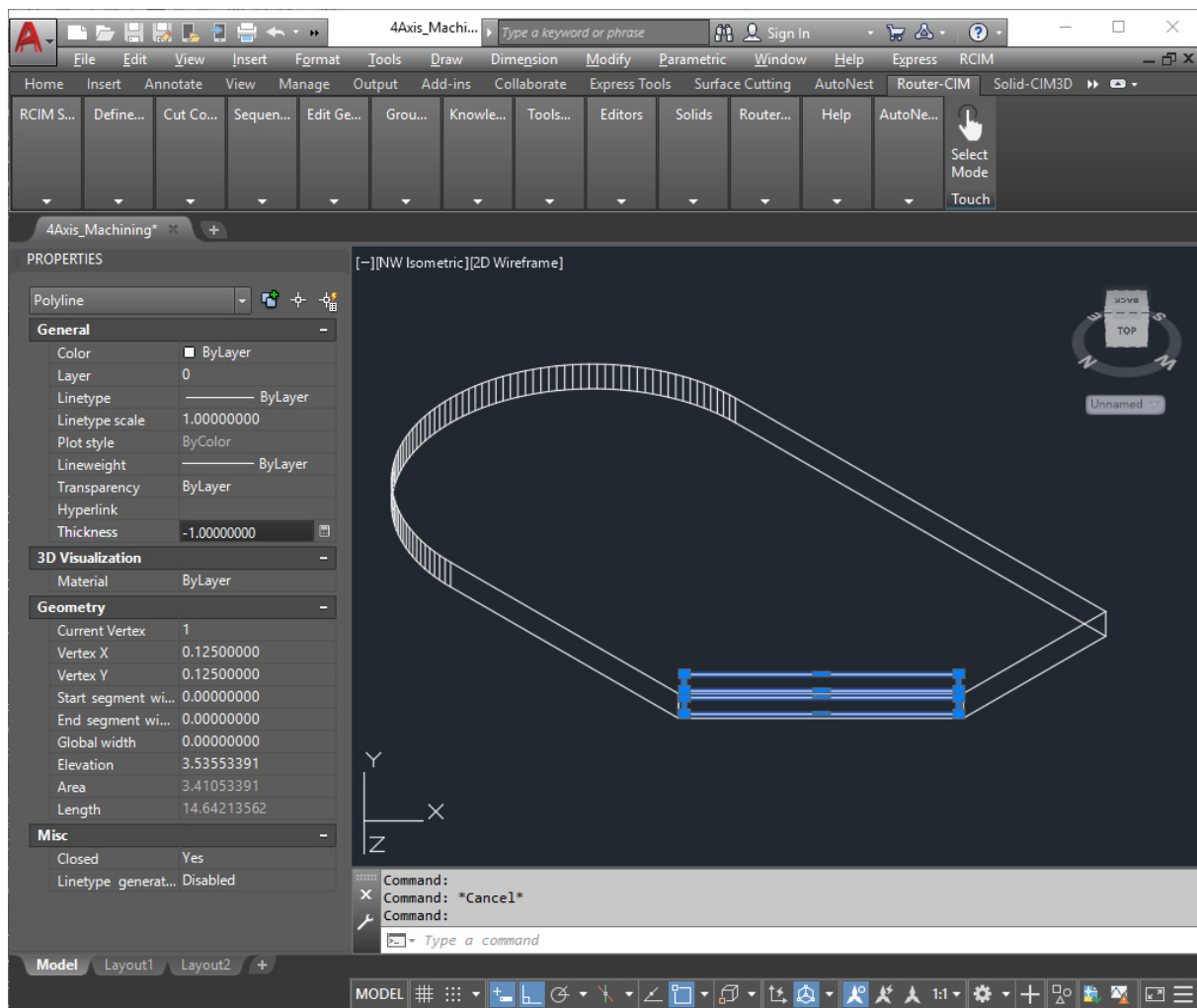
- A. If you were to face this surface, use the endpoint snap to choose the bottom right corner.
4. Choose a point on the positive portion of the Y axis.
 - A. If you were to face this surface, use the endpoint snap to choose the top left corner.
5. The UCS icon should be attached to the lower left corner of the surface and should be in line with the surface. The Z should be pointing away from the part.



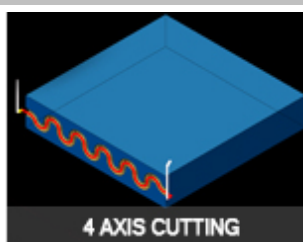
6. Draw the geometry to be cut on this surface as you normally would on the top.



7. You can now give the shape you drew thickness (always negative) so that you do not have to specify the depth of cut when making the toolpath.



4 Axis Cutting



This cycle provides continuous four-axis motion for aggregate heads and 90° router heads.

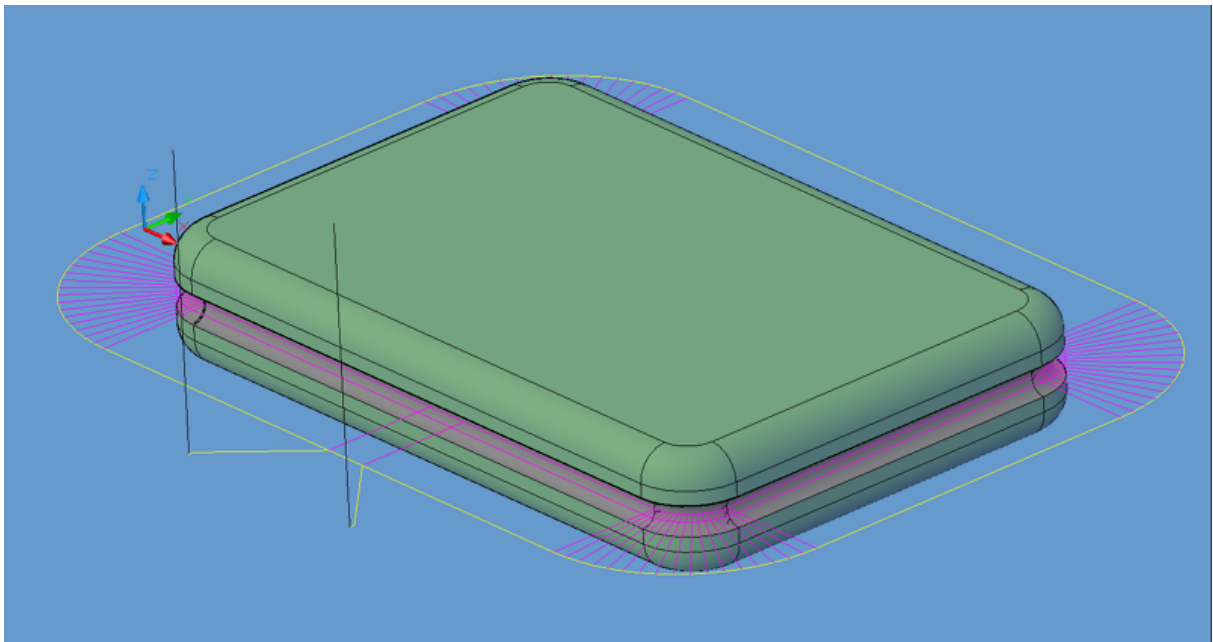
Drawing Requirements: This cycle works on standard geometry created by the Geoshape command and is edited with the Start Point Edit command. The shape Polylines must be either a two dimensional polyline in the world plane or a three dimensional shape. Two dimensional shapes on the side of the part do not need to be interpolated in the fourth axis and any or all of the existing cut cycles will work to create valid cutting motions for these shapes. If an invalid shape is selected, the cycle will notify you when you try to cut it.

The shape drawn for a fourth-axis cycle must represent the location of the material to be cut. The tool tip will extend into the cut shape by the required depth. The tool path can be shifted down in Z during the cut cycle, or the shape can be drawn at the required Z depth. For clarity, we recommend that 3D shapes be drawn at the appropriate Z depth. If a shape has thickness, this will represent the required Z shift.

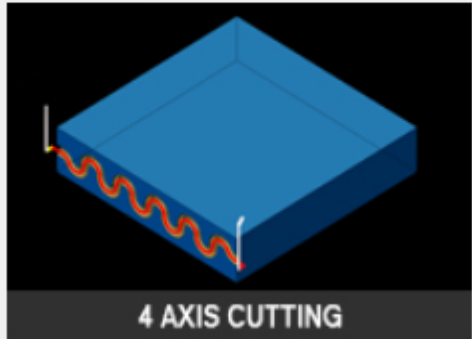
Drawing Requirements

This cycle works on standard geometry created by the Geoshape command and edited with the start point edit command. The shape polylines must be either a two dimensional polyline in the world plane or a three-dimensional shape. Two-dimensional shapes on the side of the part do not need to be interpolated in the fourth axis and any or all of the existing cut cycles will work to create cutting valid cutting motions for these shapes.

The Shape drawn for the Forth Axis Cycle must represent the material location to be cut. The tool tip will extend into the cut shape by the depth specified in the Total Cut Depth specified. The tool path can be shifted down in Z during the cut cycle by specifying the depth in the Move Shape Z field, or the shape can be drawn at the required Z depth. For clarity, we recommend that 3D shapes be drawn at the appropriate Z depth. If a shape has thickness, this will represent the required Z Shift for the Move Shape Z field.



4th Axis Cutting tool path.

Cycle Information		Status Information	
Cut Side	LH	Safety Plane	*0.25000
Cut Direction	CCW	Depth Per Pass	1.00000
Move Shape Z		Total Cut Depth	
Arc Radius In		Feedrate/Spindle Speed	
Arc Sweep		Feedrate	350.00000
Line Length		Spindle Speed	18000.00000
Line Angle		Surface FPM	NONE
Vertical Leads		Units Per Revolution	NONE
Tool Rotate Arc		Calculate	
Overlap Amount	AUTO	Before Codes	
Channel Distance		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

4th Axis Cutting parameters.

The following parameters effect the toolpath creation:

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

Select the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

Select the [Cut Direction](#) section for more information.

Move Shape Z

This value sets the Z height of the tool path. The tool path can be moved up or down with a positive or negative number in this position. Note that this is the Z position in the world coordinate system. The distance into the material horizontally is the Total Cut Depth.

Arc Radius

Lead In and out moves can have an arc lead. The radius of the arc is specified here. There can be two values placed in this position, the first being the lead-in and the second being the lead-out.

Arc Sweep

The degrees of sweep in the arcs for the leads can be specified here. There can be two values placed in this position. The first would be for the lead-in and the second would be for the lead-out.

Line Length

The length of the lead-in and lead-out line. There can be two values here, one for the lead-in and one for the lead-out.

Line Angle

The angle of the leads, specified in degrees. There can be two values here, one for the lead-in and one for the lead-out.

Vertical Leads

Specify Y for yes, or N for no. If Yes is selected, then the lead-in and lead-out moves will be in the vertical plane, instead of the XY horizontal plane.

Tool Rotate Arc

Specify Y for yes, or N for no. If Yes is selected, then the lead-in and lead-out moves will be cause the tool to rotate into the lead-in and lead-out. This is only valid for leads where Vertical Leads is set to N.

Overlap

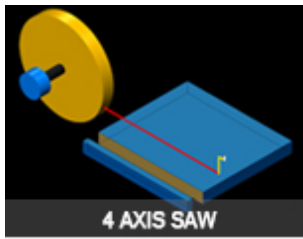
Specified as a numeric distance. The overlap is the distance the tool travels as it crossed the start point. This is useful when removing witness marks from the lead-in.

Channel Distance

If Vertical Leads is set to Y, then you can specify a Channel Distance to make the cutter ramp into the cut following the contour. Setting the distance to a small number will cause a steep, sharp ramp, and setting it to a larger number will make a longer, more gradual ramp. There can be two values here, one for the lead-in and one for the lead-out.

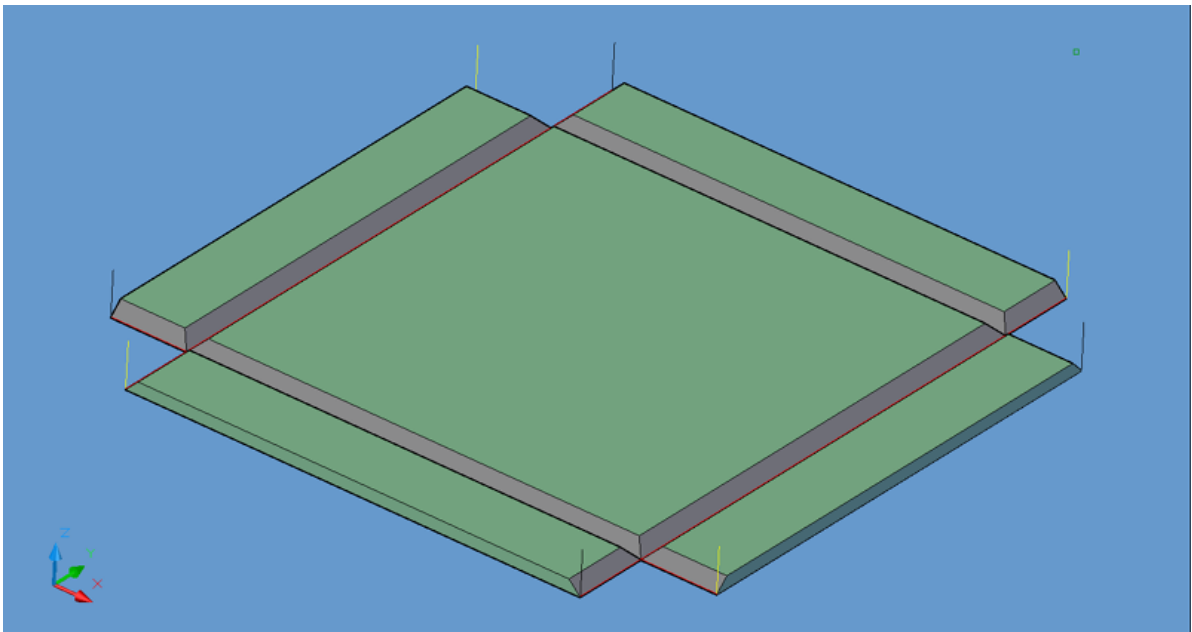
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

4 Axis Saw

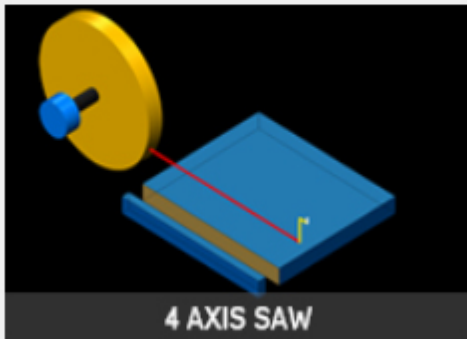


This cycle is set up much the same way as a normal straight cut, except use the tool number for the four-axis saw head. Router-CIM will interpret the axis geometry and all moves required to complete the cycle. The Control Panel settings for this cycle are the same as for a 2D cycle.

- The 4 Axis Safe field has no function in this cycle.
- The Tool Length field is used to determine the distance from the center of the saw to the center of the blade.
- Cutter Compensation is available in this cycle.



4 Axis Saw tool paths.

Cycle Information		Status Information	
Saw Offset	0.0	Safety Plane	*0.25000
Cut Side	LH	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Lead In	N	Feedrate/Spindle Speed	
Lead Out	N	Feedrate	350.00000
Lead Size	0.0	Spindle Speed	18000.00000
Lead Feed		Surface FPM	NONE
		Units Per Revolution	NONE
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

4th Axis Saw parameters.

The following parameters effect the toolpath creation:

Saw Offset

The saw offset is the amount the toolpath is offset from the original geometry or Geoshape.

Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

For a saw, you might want the tool path set on top of your geometry with no offset (use 0) or offset by some amount relative to your saw spindle or aggregate.

You may substitute the parameters here for numeric values to suit your particular cutting needs.

See [Offset Dim](#) for information that is relevant to Saw Offset as well.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

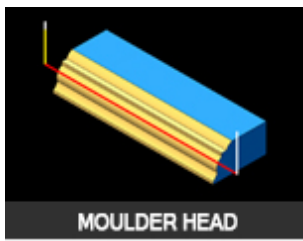
Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

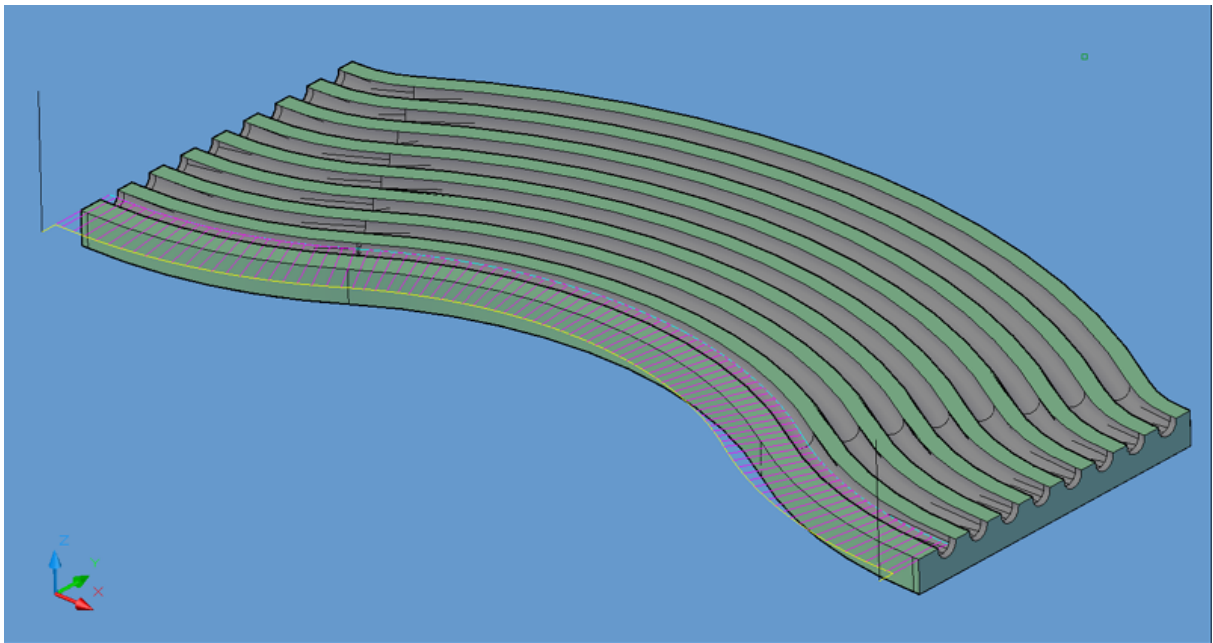
4 Axis Moulder Head



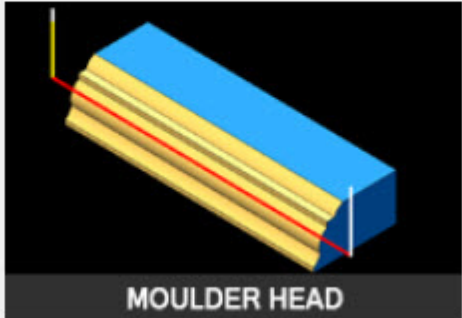
This cycle provides continuous four-axis motion for aggregate moulder heads and fixed moulder heads.

Drawing Requirements: This cycle works on standard geometry created by the Geoshape command and is edited with the Start Point Edit command. The shape Polylines must be either a two dimensional polyline in the world plane or a three dimensional shape. Two dimensional shapes on the side of the part do not need to be interpolated in the fourth axis and any or all of the existing cut cycles will work to create valid cutting motions for these shapes. If an invalid shape is selected, the cycle will notify you when you try to cut it.

The shape drawn for a fourth-axis cycle must represent the location of the material to be cut. The tool tip will extend into the cut shape by the required depth. The tool path can be shifted down in Z during the cut cycle, or the shape can be drawn at the required Z depth. For clarity, we recommend that 3D shapes be drawn at the appropriate Z depth. If a shape has thickness, this will represent the required Z shift.



Moulder Head tool path.

Cycle Information		Status Information	
Cut Side	LH	Safety Plane	*0.25000
Cut Direction	CCW	Depth Per Pass	1.00000
Move Shape Z		Total Cut Depth	
Arc Radius In		Feedrate/Spindle Speed	
Arc Sweep		Feedrate	1000.00000
Line Length		Spindle Speed	18000.00000
Line Angle		Surface FPM	NONE
Vertical Leads		Units Per Revolution	NONE
Tool Rotate Arc		Ramp Feedrate	
Overlap		Calculate	
Channel Distance		Before Codes	
Over-Wind	N	After Codes	
Lead Feed	0.5	Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Moulder Head parameters.

The following parameters effect the toolpath creation:

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

Select the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any

closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

Select the [Cut Direction](#) section for more information.

Move Shape Z

This value sets the Z height of the tool path. The tool path can be moved up or down with a positive or negative number in this position. Note that this is the Z position in the world coordinate system. The distance into the material horizontally is the Total Cut Depth.

Arc Radius

Lead In and out moves can have an arc lead. The radius of the arc is specified here. There can be two values placed in this position, the first being the lead-in and the second being the lead-out.

Arc Sweep

The degrees of sweep in the arcs for the leads can be specified here. There can be two values placed in this position. The first would be for the lead-in and the second would be for the lead-out.

Line Length

The length of the lead-in and lead-out line. There can be two values here, one for the lead-in and one for the lead-out.

Line Angle

The angle of the leads, specified in degrees. There can be two values here, one for the lead-in and one for the lead-out.

Vertical Leads

Specify Y for yes, or N for no. If Yes is selected, then the lead-in and lead-out moves will be in the vertical plane, instead of the XY horizontal plane.

Tool Rotate Arc

Specify Y for yes, or N for no. If Yes is selected, then the lead-in and lead-out moves will be cause the tool to rotate into the lead-in and lead-out. This is only valid for leads where Vertical Leads is set to N.

Overlap

Specified as a numeric distance. The overlap is the distance the tool travels as it crossed the start point. This is useful when removing witness marks from the lead-in.

Channel Distance

If Vertical Leads is set to Y, then you can specify a Channel Distance to make the cutter ramp into the cut following the contour. Setting the distance to a small number will cause a steep, sharp ramp, and setting it to a larger number will make a longer, more gradual ramp. There can be two values here, one for the lead-in and one for the lead-out.

Over-Wind

Specify Y for yes, or N for no. If Yes is selected, allows the moulder head cycle to travel +/-5 degrees past 0 or 360 degree rotation. Only applicable with certain CNC machines.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

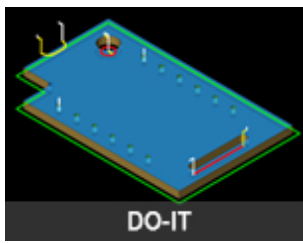
Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

DO-IT



DO-IT is automatic tool and cycle selection, accomplished using a knowledge base you define according to your individual cutting needs. This knowledge base and Layer to Knowledge level cutting is the basis for the Automation Suite inside of Router-CIM.

Using DO-IT you can place geometry on specific layers, and then associate those layers to specific cutting knowledges via a "DOIT File" or association file that stores the relationships of the layer to knowledge cutting. DO-IT will automatically search the file for the associations and then if the layer is present and the knowledge is present in the current drawing, it will place tool paths on the part according to the associations found.

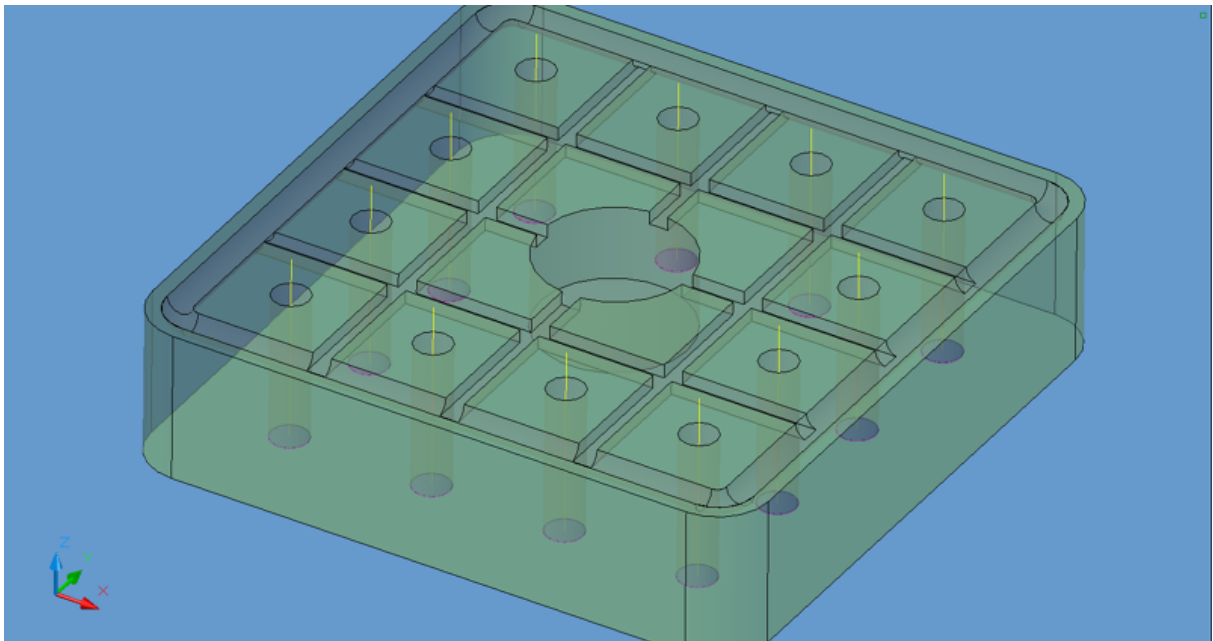
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**


Tap-Matic



The Tap Matic cycle is used to program a drill type motion for a Tap Matic head fitted to a spindle on the machine. The Tap Matic head requires a feedrate for the motion down into the hole, and another feedrate on the way back up out of the hole.

Router-CIM provides for the separate feedrates in its drill motions using this cycle so that the Tap Matic head can be used. The geometry needed is simply a circle, geoshaped, and otherwise the cycle works the same as Drill Motions.



Cycle Information	Status Information
Index Speed <input type="text"/>	Safety Plane <input type="text" value="0.25000"/>
Retract Feed <input type="text"/>	Depth Per Pass <input type="text" value="1.00000"/>
	Total Cut Depth <input type="text"/>
	Feedrate/Spindle Speed
	Feedrate <input type="text" value="350.00000"/>
	Spindle Speed <input type="text" value="18000.00000"/>
	Surface FPM <input type="text" value="NONE"/>
	Units Per Revolution <input type="text" value="NONE"/>
	<input type="button" value="Calculate"/>
	Before Codes <input type="text"/>
	After Codes <input type="text"/>
	Oscillation Amount <input type="text" value="0.00000"/>
	Sort By Rank # <input type="text"/>
	
	<input type="button" value="Reset Cycle Settings to Default"/>

The following parameters effect the toolpath creation:

Index Speed

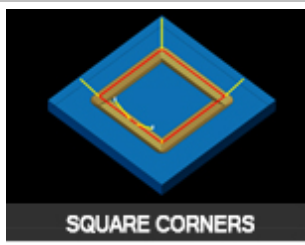
This is the fastest speed that you want the machine to achieve between the drilled holes. This feedrate will take place of the rapid traverse move between cuts and during the retract of the hole. If the machine can make a fast linear move between the cuts, usually this will reduce the overall cycle time of the drill moves.

Retract Feed

This cycle has a separate feedrate for the lead-out. You should specify the lead-out feedrate in this position.

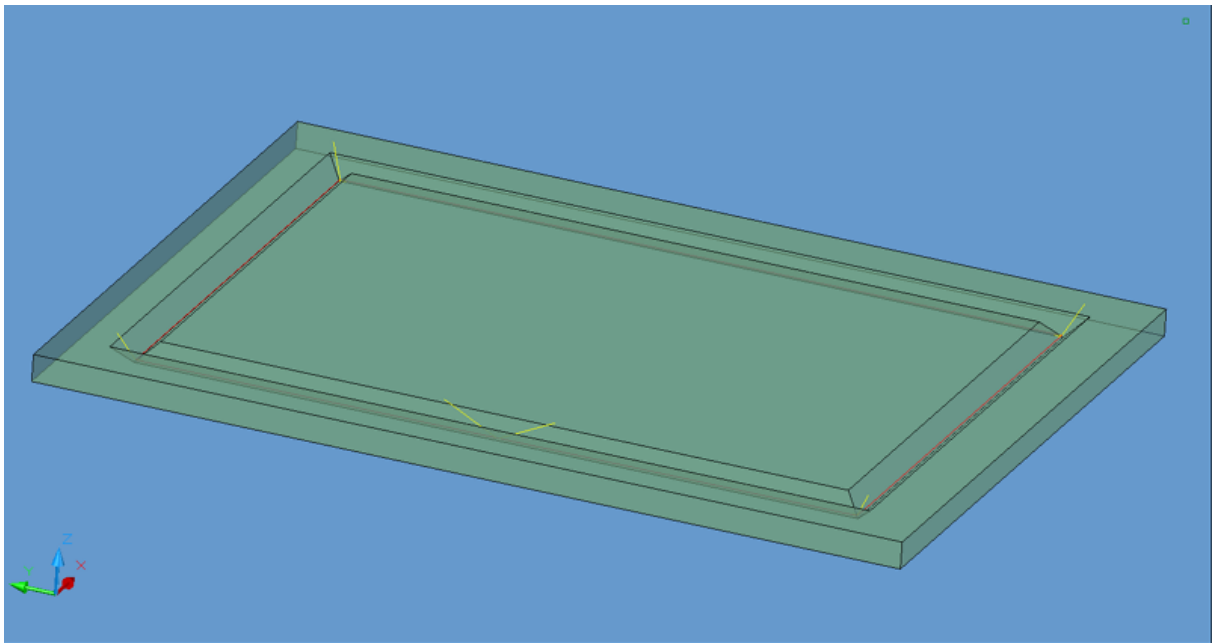
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Square Corners

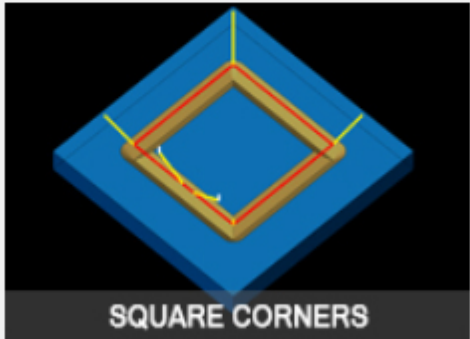


The Square Corners cycle is most often used when making solid doors (such as MDF single piece doors) and the desired effect is to have a squared off corner cut with an angled tool so that there is a somewhat sharp edge on the inside of the profile of the door.

This is accomplished by moving the angled tool up and down in each corner, using the angle of the tool to make a squared off edge.



Square Corner tool path

Cycle Information		Status Information	
Offset Dim	OFFSZ	Safety Plane	*0.25000
Cut Side	INSIDE	Depth Per Pass	1.00000
Cut Direction	CCW	Total Cut Depth	
Round Corners	N	Feederate/Spindle Speed	
Lead In	N	Feederate	350.00000
Lead Out	N	Spindle Speed	18000.00000
Lead Size	0.0	Surface FPM	NONE
Lead Angle	N	Units Per Revolution	NONE
Lead Ratio		Calculate	
Lead Feed		Before Codes	
Crowsfeet	Y	After Codes	
Crowsize	0.2	Oscillation Amount	0.00000
Tool Angle		Sort By Rank #	
Angle Length			
		Reset Cycle Settings to Default	

Square Corner parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the [Lead Angle](#) section for more information.

Lead Ratio

Lead Ratio determines the angle of the ramp in Z during the lead in and lead out. You can specify the Lead Ratio as a number that reflects the percentage of the angle from its default. That means that if you want a lead that is twice the normal ramp length (shallower angle) enter 2. If you want a lead that is steeper than the default, enter .5.

See the [Lead Ratio](#) section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

Crowsfeet

Possible answers here are Y for yes, or N for no. This parameters specifies whether or not to make tiny corner clean up moves at the bottom of the angled corner cut.

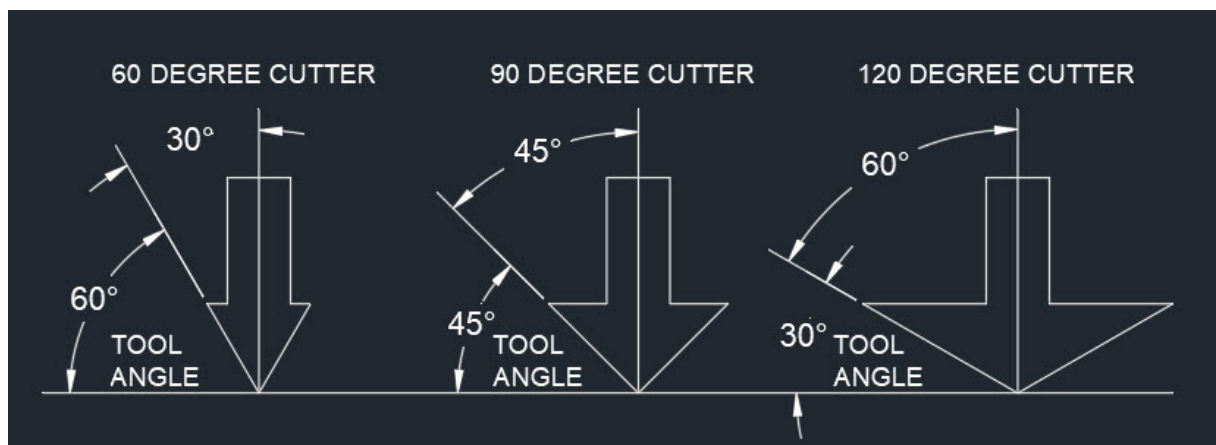
Crowsize

Setting this to a numeric value will allow you to control the size of the crowsfeet made at the bottom of the angled corner cut.

Crowsfeet must be set to Y in order for this setting to have any effect.

Tool Angle

Placing the tool angle in this parameter allows the square corner move to be at the correct angle to match the tool and the geometry. Tool angle is calculated from center of tool in one direction.

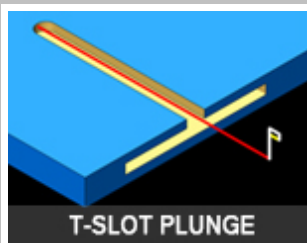


Angle Length

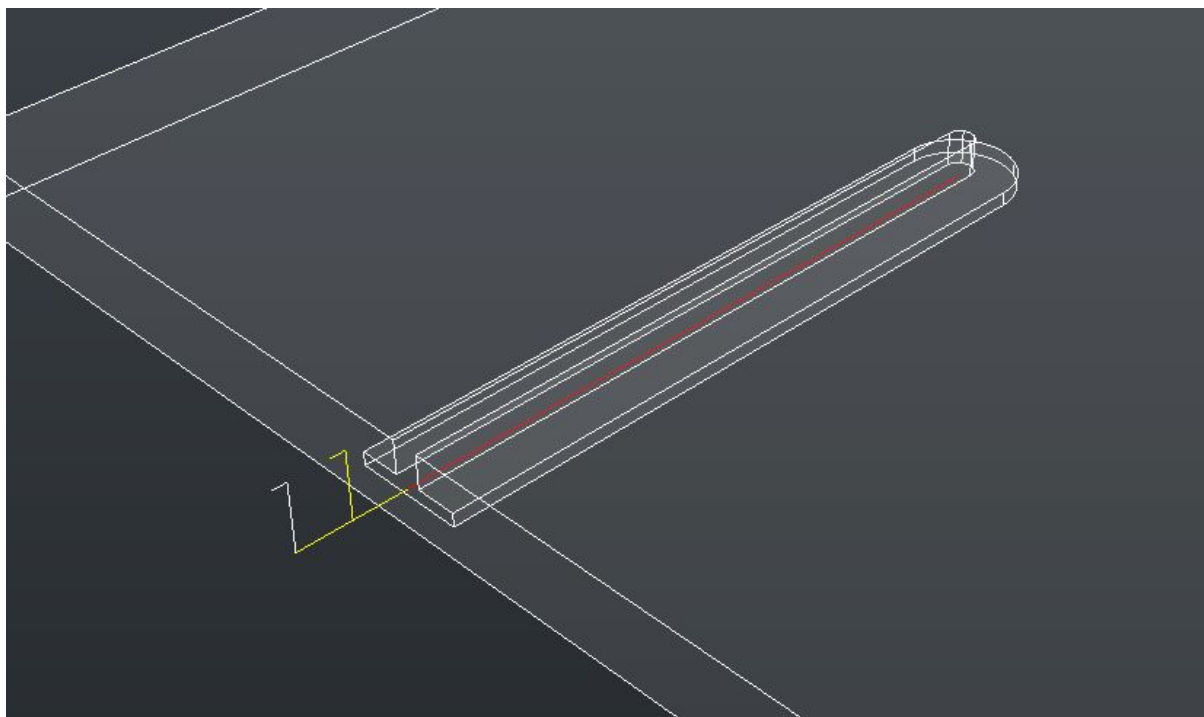
This is a numeric value for the total length of the square corner move.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

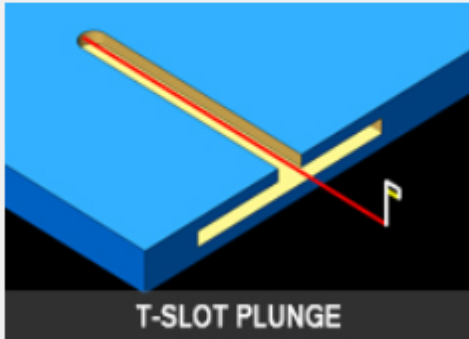
T-Slot Plunge



The T-Slot Plunge cutting cycle gives you the ability to follow an open piece of geometry that will trace the cut path back along its original path before it retracts to exit the cut.



T-Slot Plunge Tool Path

Cycle Information	Status Information
XY Stock Allowance <input type="text"/>	Safety Plane <input type="text" value="0.25000"/>
Z Stock Allowance <input type="text"/>	Depth Per Pass <input type="text" value="1.00000"/>
Lead Feed <input type="text"/>	Total Cut Depth <input type="text"/>
	Feedrate/Spindle Speed
	Feedrate <input type="text" value="350.00000"/>
	Spindle Speed <input type="text" value="18000.00000"/>
	Surface FPM <input type="text" value="NONE"/>
	Units Per Revolution <input type="text" value="NONE"/>
	<input type="button" value="Calculate"/>
	Before Codes <input type="text"/>
	After Codes <input type="text"/>
	Oscillation Amount <input type="text" value="0.00000"/>
	Sort By Rank # <input type="text"/>
	 <p>T-SLOT PLUNGE</p>
	<input type="button" value="Reset Cycle Settings to Default"/>

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape.

The value set by default (0) is a setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed unless you want to force an offset for the tool path.

Cutter Compensation needs to be set to 'NO' for this cycle.

See [Offset Dim](#) for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the [Cut Side](#) section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the [Cut Direction](#) section for more information.

Round Corners

The value is not applicable to a T-Slot Plunge function.

See the [Round Corners](#) section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. Depending on if the geometry is already extended past the edge of the part will determine if a Lead-In will need to be defined.

In the picture above, the Lead-In was set to "LNTLI"

See the [Lead-In](#) section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. Depending on if the geometry is already extended past the edge of the part will determine if a Lead-Out will need to be defined.

In the picture above, the Lead-In was set to "LNTLO"

See the [Lead-Out](#) section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the [Lead-Size](#) section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the [Lead Angle](#) section for more information.

XY Stock Allowance

The value is not applicable to a T-Slot Plunge function.

See [XY Stock Allowance](#) for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See [Z Stock Allowance](#) for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).

Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the [Lead Feed](#) section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Advanced Machining Cycles (Optional)

Note: The Advanced Machining Cycles are an optional add-in that will give you additional cutting cycles with advanced machining capability. Contact sales@cim-tech.com for more information.


When installed, the additional cutting cycles will be available through the cutting cycles pages of the Router-CIM control panel.

Each additional cutting cycle has specific cycle parameters that are discussed in the following section.

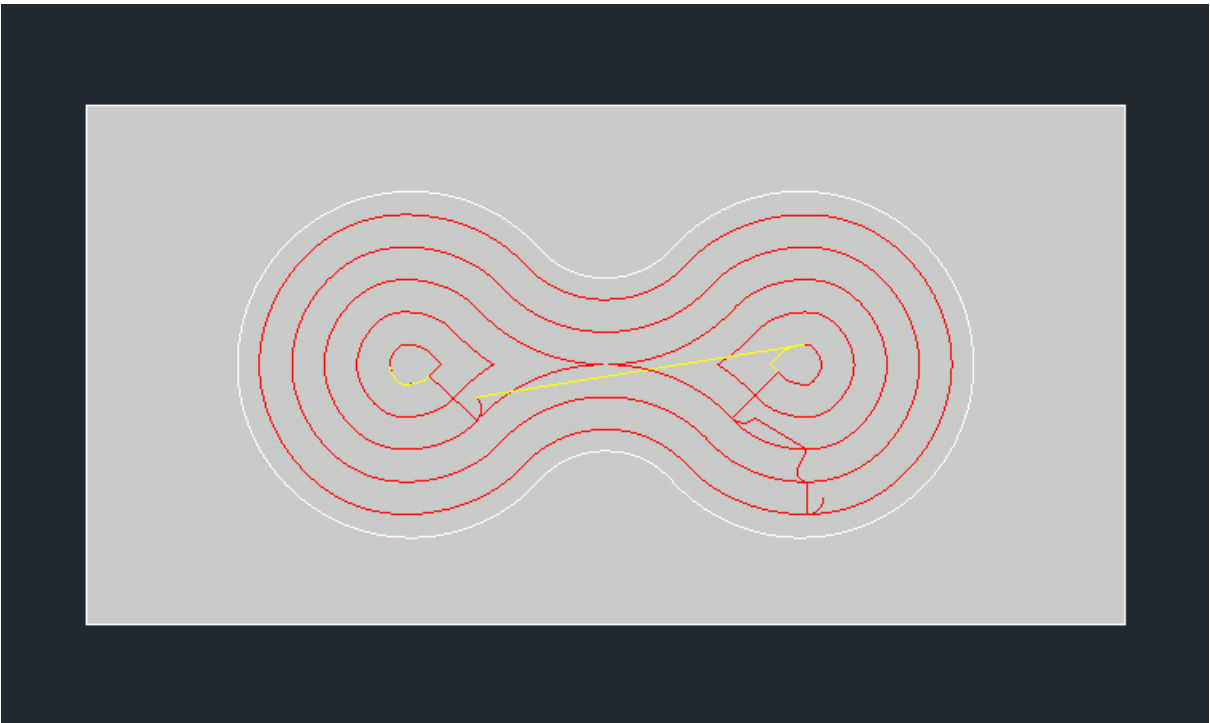
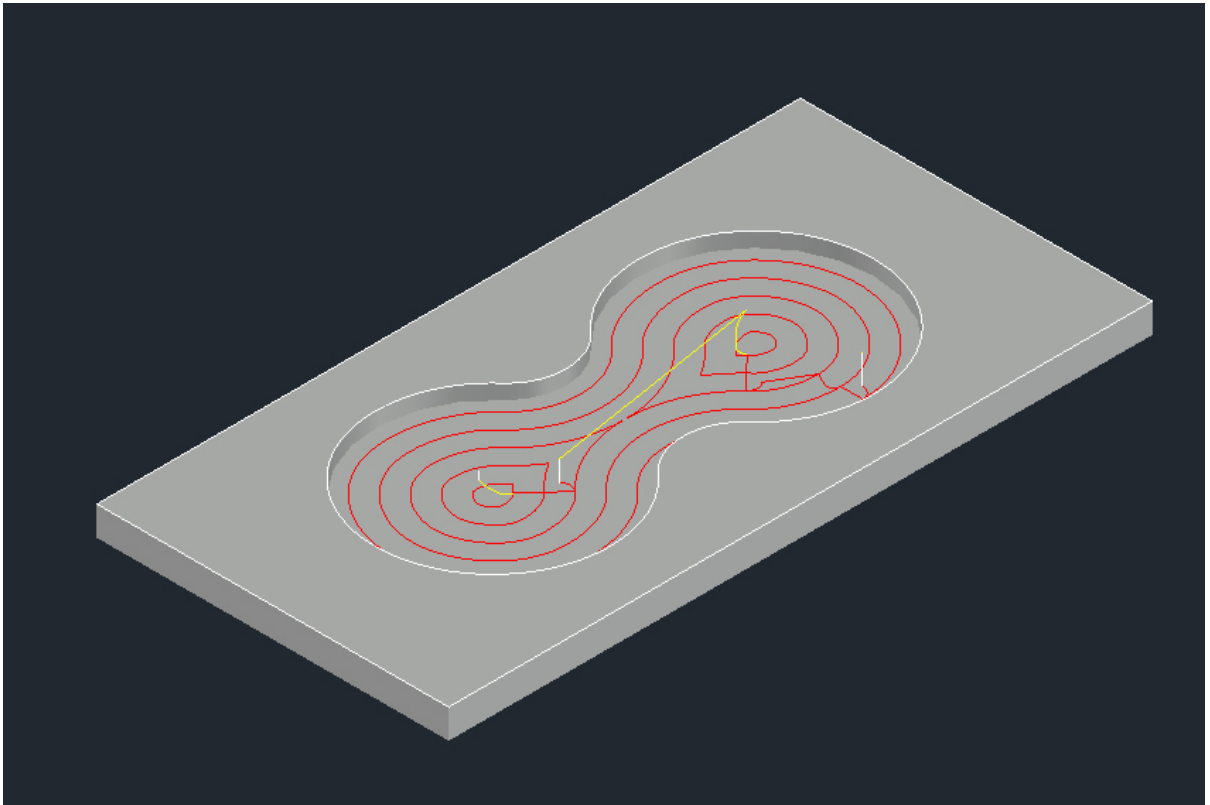
Available Advanced Machining Cycles:




2D Pocket

	<p>Advanced 2D Pocketing uses advanced algorithms based on the wire frame input geometry and contains many options to control the creation of the tool path than the other pocketing cycles. This method allows for more complex geometry to be handed to the tool path generator.</p> <p>The Advanced 2D Pocketing utilizes numerous parameters to generate the most efficient toolpath around a pocket. There are parameters to control the lead in motion, stepover distance, stepover type, and more.</p> <p>The Advanced 2D Pocketing will start at a point inside the pocket and create offset tool paths, moving toward the outside of the pocket.</p>
---	---

Note: The Advanced Machining Cycles are an optional add-in that will give you additional cutting cycles with advanced machining capability. Contact sales@cim-tech.com for more information.



Cycle Information		Status Information	
Stock Offset	0.0000	Safety Plane	*0.25000
Stepover	TW75	Depth Per Pass	1.00000
Stepover Type	Retract_to_Clearance_Area	Total Cut Depth	
Stay Down	Y	Feedrate/Spindle Speed	
Clean Corners	N	Feedrate	1000.00000
Cut Direction	Climb	Spindle Speed	18000.00000
Lead Feedrate	0.0	Surface FPM	NONE
Lead-In Type	Automatic	Units Per Revolution	NONE
Lead-In Angle	0.0	Ramp Feedrate	
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		Reset Cycle Settings to Default	

Advanced-2D-Pocket-Parameters

The following parameters effect the toolpath creation:

Stock Offset <0.0000>

The value entered here will be added to the cut offset to provide material left for a clean up pass on the pocket with a separate tool.

Stepover <TW75>

This value defines the stepover made after each offset.

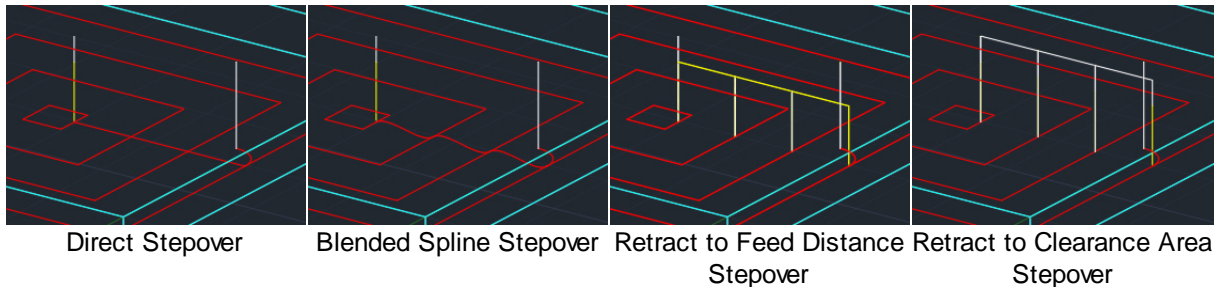
- 1) Default of TW75 will step over by 75% of the tool diameter
- 2) Number followed by the % sign. This needs to be entered as a percent number such as 25% or 50%. This will stepover the percentage of the tool diameter

3) Number only. You can identify a specific stepover offset by just putting in the value as long as it does not exceed the diameter of the tool. Value of 0.25 would stepover 0.25 regardless of the tool diameter.

Stepover Type <Retract-to-Clearance-Area>

There are different methods for moving from one step to the next step in the cutting cycle. This parameter defines the type of connection move between the offset cuts of each pass.

- 1) Direct - This is a straight line connection between passes on the shortest way without any retracting movements keeping the cutting tool at the current Z level for the pass.
- 2) Blend Spline - Tangential arcs connecting between the passes
- 3) Retract to Feed Distance - The cutting tool will retract to the feed distance to material variable after each stepover offset pass before moving to the next stepover offset.
- 4) Retract to Clearance Area - The cutting tool will retract to the safety plane after each stepover offset pass before moving to the next stepover offset.



Stay Down <Y>

If this parameter is set to <Y>, it will keep the tool down while in the pocket to continue the shape. If it is set to <N>, it will keep allow the tool to pick up and move to another area of the pocket to continue the shape.

Clean Corners <N>

When set to "Y", the toolpath will be extended into corners if the stepover does not reach.

Cut Direction <Climb>

The value represents if the cutting cycle will be doing Climb (CCW) milling or if you want the cutting cycle to do Conventional (CW) milling when pocketing inside geometry.

Lead Feedrate <0.0>

This sets lead-in and lead-out feed rates. The default is 50%. Setting this parameter to any number will override the lead-in feedrate to the number specified. Setting this parameter to a percentage, 25% (must include the % symbol) will adjust the lead-in based on the feedrate identified by the percentage defined on CNC machines with this capability.

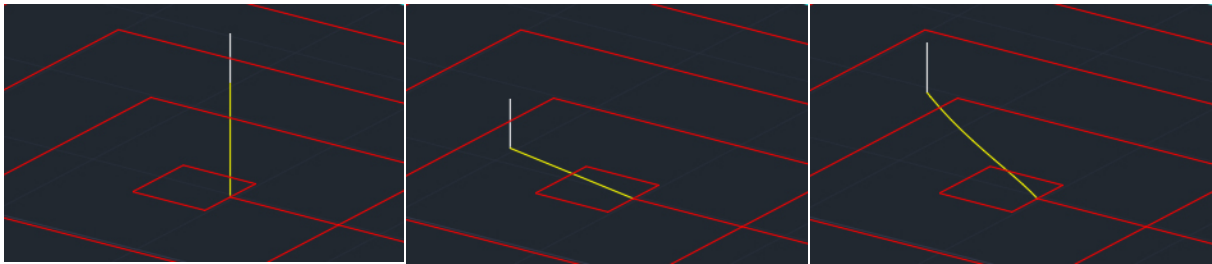
Lead-In Type <Automatic>

This parameter defines the type of entry move the toolpath will make when making its initial approach.

The possible options are:

- 1) Automatic – The approach type will be picked automatically to avoid collision
- 2) Line – The ramp move is a slanted line when a Lead-In Angle is used
- 3) Helical – The ramp move is helical
- 4) ZigZag – The ramp move is alternating slanted linear moves in opposite directions

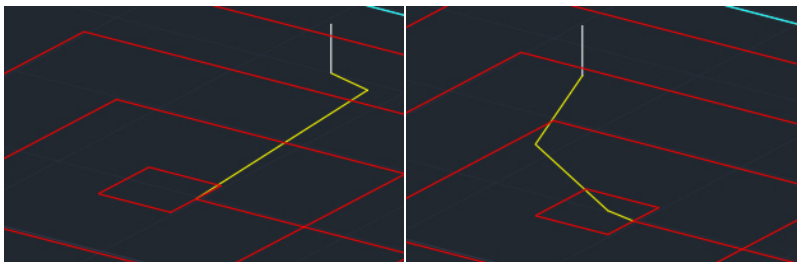
5) Profile – The ramp follows the toolpath contour shape while gradually plunging



Lead-in Line No Angle

Lead-In Line 30 Degree Angle

Lead-In Helical 30 Degree Angle



Lead-In ZigZag 30 Degree Angle

Lead-In Profile 30 Degree Angle

Lead-In Angle <0.0>

The parameter will need a numeric value defining the degrees of the ramp such as 30 or 45.

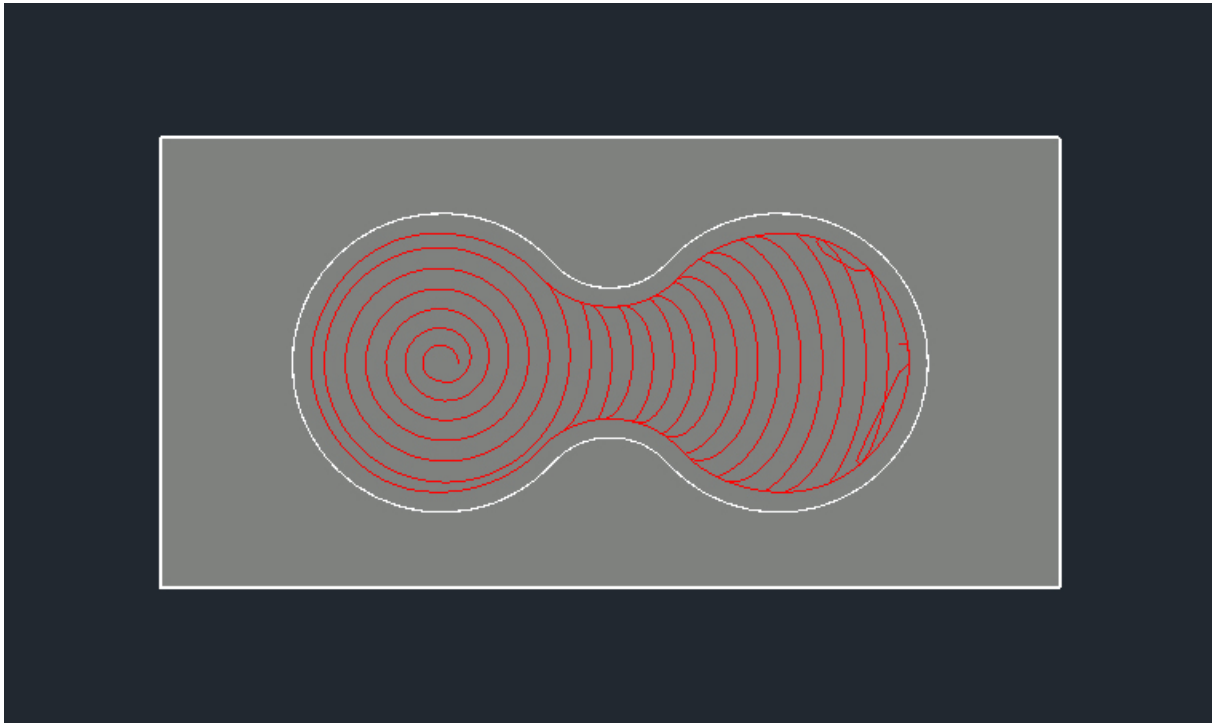
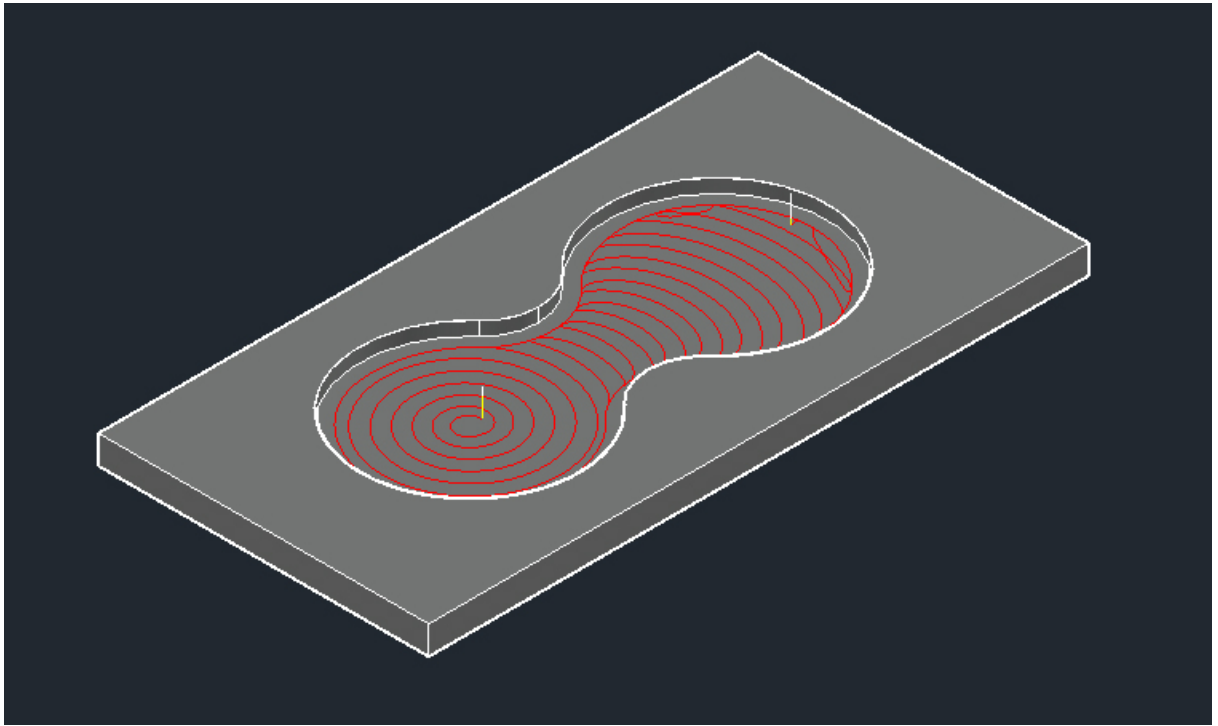
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

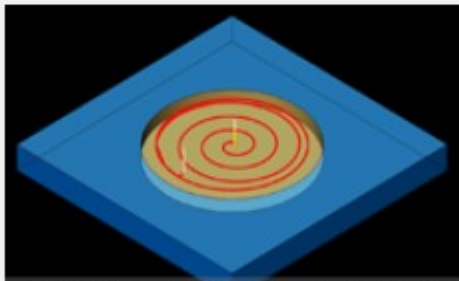
Advanced Spiral Pocketing



This version of Advanced Pocketing uses adaptive roughing algorithms based on the wire frame input geometry and contains many options to control the creation of the tool path than the other pocketing cycles. This adaptive strategy ensures that the cutting conditions remain almost constant by measuring the engagement volume of the tool with material and gradually removing material off the remaining stock. It guarantees stables load on the tool allowing for increased material removal rate at higher feed rates and reduces overall machining time.

Note: The Advanced Machining Cycles are an optional add-in that will give you additional cutting cycles with advanced machining capability. Contact sales@cim-tech.com for more information.



Cycle Information		Status Information	
Stock Offset	0.0000	Safety Plane	*0.25000
Stepover	TW75	Depth Per Pass	1.00000
Cut Direction	Climb	Total Cut Depth	
Lead Feedrate	0.0000	Feedrate/Spindle Speed	
Lead-In Type	Automatic	Feedrate	1000.00000
Lead-In Angle	0.0	Spindle Speed	18000.00000
		Surface FPM	NONE
		Units Per Revolution	NONE
		Ramp Feedrate	
		Calculate	
		Before Codes	
		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	
			
		ADVANCED SPIRAL POCKETING	
		Reset Cycle Settings to Default	

The following parameters effect the toolpath creation:

Stock Offset <0.0000>

The value entered here will be added to the cut offset to provide material left for a clean up pass on the pocket with a separate tool.

Stepover <TW75>

This value defines the stepover made after each offset.

- 1) Default of TW75 will step over by 75% of the tool diameter
- 2) Number followed by the % sign. This needs to be entered as a percent number such as 25% or 50%. This will stepover the percentage of the tool diameter

3) Number only. You can identify a specific stepover offset by just putting in the value as long as it does not exceed the diameter of the tool. Value of 0.25 would stepover 0.25 regardless of the tool diameter.

Cut Direction <Climb>

The value represents if the cutting cycle will be doing Climb (CCW) milling or if you want the cutting cycle to do Conventional (CW) milling when pocketing inside geometry.

Lead Feedrate <0.0>

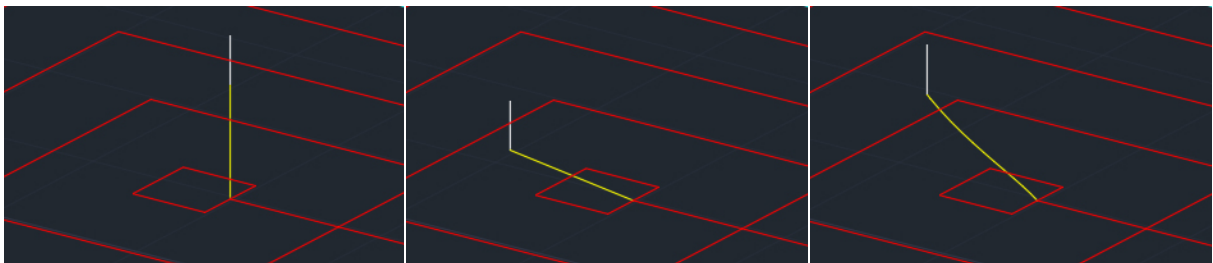
This sets lead-in and lead-out feed rates. The default is 50%. Setting this parameter to any number will override the lead-in feedrate to the number specified. Setting this parameter to a percentage, 25% (must include the % symbol) will adjust the lead-in based on the feedrate identified by the percentage defined on CNC machines with this capability.

Lead-In Type <Automatic>

This parameter defines the type of entry move the toolpath will make when making its initial approach.

The possible options are:

- 1) Automatic – The approach type will be picked automatically to avoid collision
- 2) Line – The ramp move is a slanted line when a Lead-In Angle is used
- 3) Helical – The ramp move is helical



Lead-in Line No Angle

Lead-In Line 30 Degree Angle

Lead-In Helical 30 Degree Angle

Lead-In Angle <0.0>

The parameter will need a numeric value defining the degrees of the ramp such as 30 or 45.

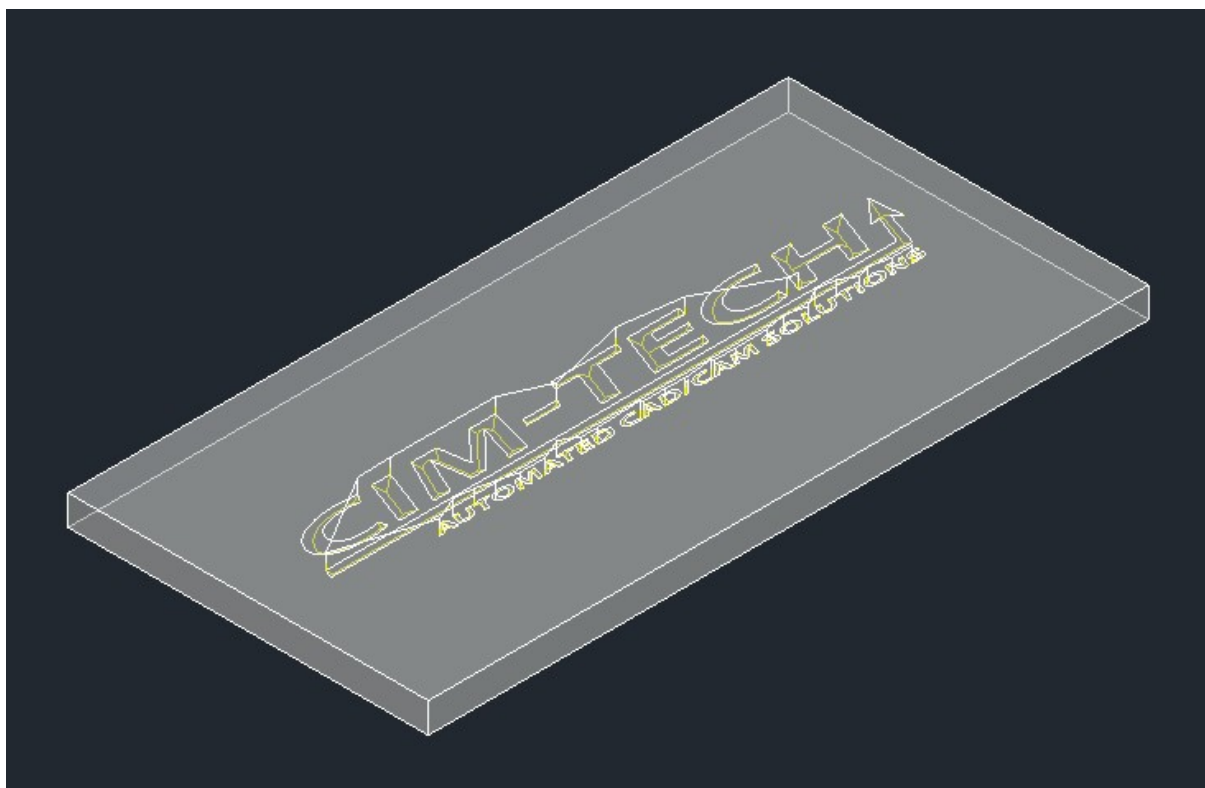
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

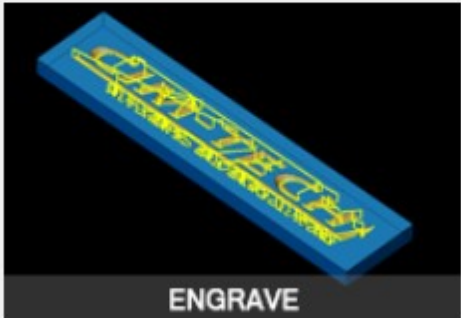
Engrave



This cycle uses advanced algorithms to automatically engrave 2D letters/geometry. This calculation has built-in functionality to modify the input geometry in order to cover certain intersections

Note: The Advanced Machining Cycles are an optional add-in that will give you additional cutting cycles with advanced machining capability. Contact sales@cim-tech.com for more information.



Cycle Information		Status Information	
Stock Offset	<input type="text" value="0.0000"/>	Safety Plane	<input type="text" value="*0.25000"/>
Lead Feedrate	<input type="text" value="0.0000"/>	Depth Per Pass	<input type="text" value="1.00000"/>
		Total Cut Depth	<input type="text"/>
		Feedrate/Spindle Speed	
		Feedrate	<input type="text" value="1000.00000"/>
		Spindle Speed	<input type="text" value="18000.00000"/>
		Surface FPM	<input type="text" value="NONE"/>
		Units Per Revolution	<input type="text" value="NONE"/>
		Ramp Feedrate	<input type="text"/>
		<input type="button" value="Calculate"/>	
		Before Codes	<input type="text"/>
		After Codes	<input type="text"/>
		Oscillation Amount	<input type="text" value="0.00000"/>
		Sort By Rank #	<input type="text"/>
			
		<input type="button" value="ENGRAVE"/>	
		<input type="button" value="Reset Cycle Settings to Default"/>	

The following parameters effect the toolpath creation:

Stock Offset <0.0000>

The value entered here will be added to the cut offset to provide material left for a clean up pass on the pocket with a separate tool.

Lead Feedrate <0.0>

This sets lead-in and lead-out feed rates. The default is 50%. Setting this parameter to any number will override the lead-in feedrate to the number specified. Setting this parameter to a percentage, 25% (must include the % symbol) will adjust the lead-in based on the feedrate identified by the percentage defined on CNC machines with this capability.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

Cycle Parameters

There are many parameters associated with the cutting cycles, but a majority of them share some parameters in common. This section will focus on some of the shared parameters and offer a more detailed explanation of them.

Offset Dim

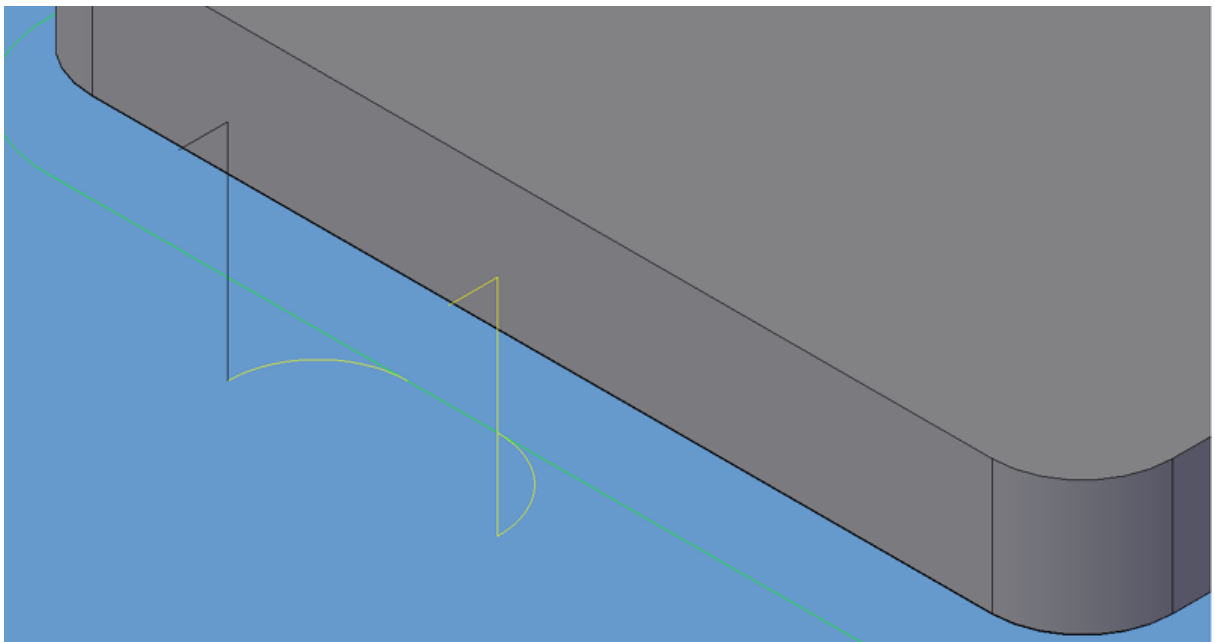
Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape.

Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit your particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.



The options selected from the Options Box by default are:

firstxy xycutloc -- This is a macro which will allow Router-CIM to perform the normal offset depending on various settings on the cut such as cutter compensation.

0.0 -- This will set the offset amount to 0 and leave the toolpath directly on top of the geoshaped entity.

offsz -- This is also a Router-CIM macro, which allows the offset to be handled by Router-CIM.

!*tr* -- A Router-CIM macro which will set the offset to the tools radius.

0 -- This will set the offset amount to 0 and leave the toolpath directly on top of the geoshaped entity.

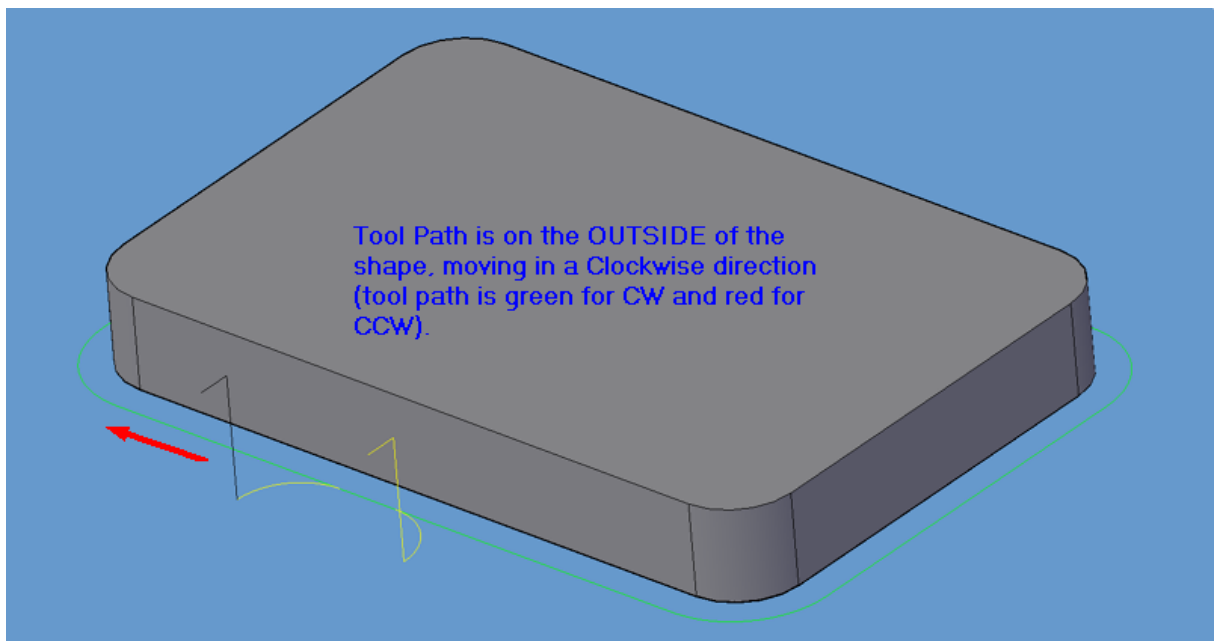
Cut Side

Cut Side

Cut Side is the side of the geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

Outside

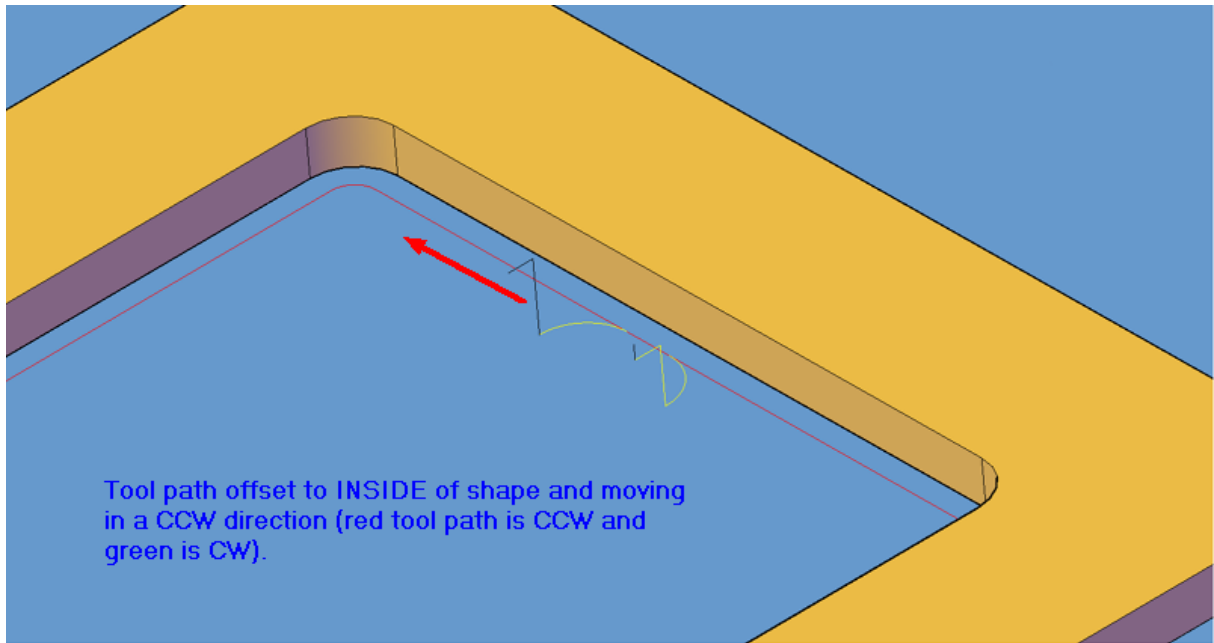
Places the tool path on the Outside of a closed shape as determined by the Geoshape command.



Tool path offset to outside.

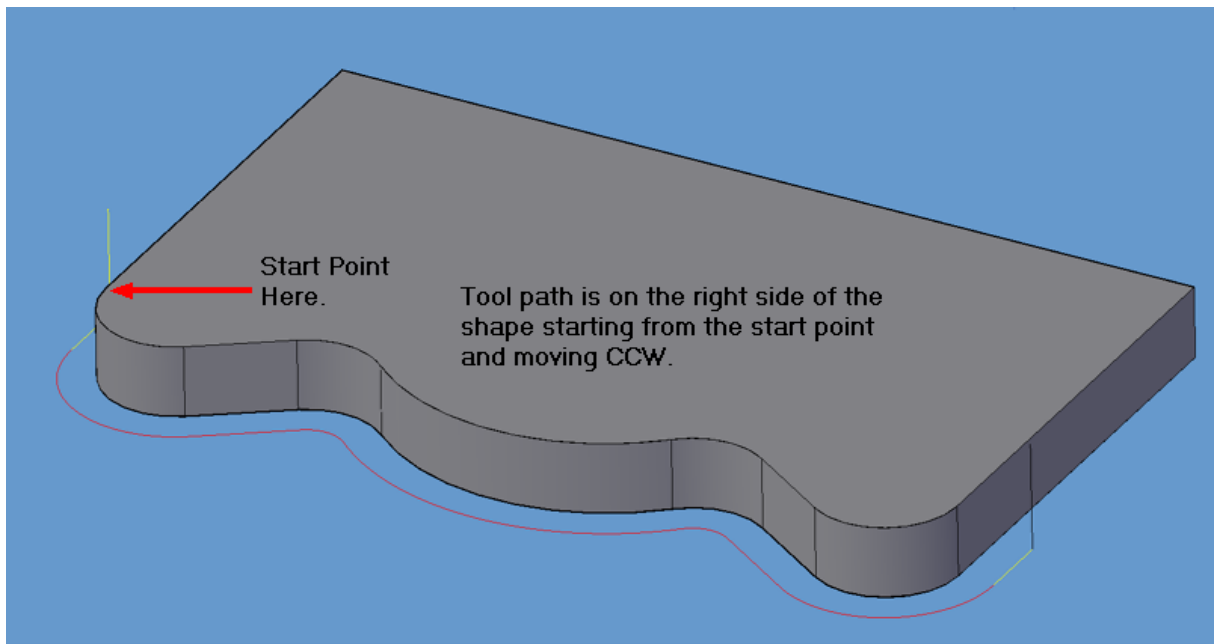
Inside

Places the tool path on the Inside of a closed shape as determined by the Geoshape command.



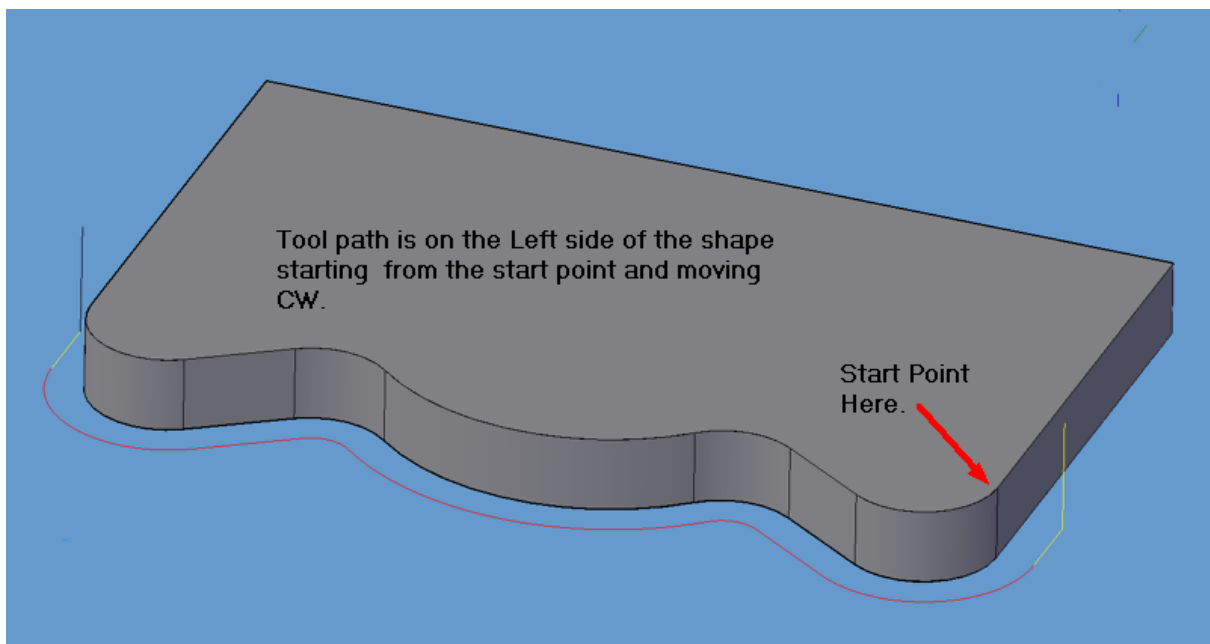
RH

Places the tool path on the right side of a closed shape as determined by the Geoshape command.



LH

Places the tool path on the Left side of a closed shape as determined by the Geoshape command.



The drawing below shows different Cut Side/Cut Direction combinations based on the shape direction, to assist you in choosing a Start Point.



It is very common when cutting open shapes that you will need to change the side of the cut. Clicking the option box (small square to the right) allows you to choose the correct option: Inside, Outside, LH, and RH.

Inside and Outside work best on closed shapes; LH and RH work best on open shapes. To check the direction of an open shape use the AutoCAD command /pedit/Edit Vertex.

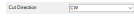
The best way to cut an open shape is to:

1. Select the Plunge-O Line Leads cycle.
2. Set the Cut Direction to CCW.
3. Set the Cut Side to RH or LH, depending on which side you want to cut.

OR

1. Use the Right Hand Cut cycle to cut on the right side of an open shape
2. Use the Left Hand Cut cycle to cut on the left side of an open shape

Cut Direction



The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

CW

This is the setting for Clockwise direction. CW tool paths in Router-CIM will appear in Green by default.

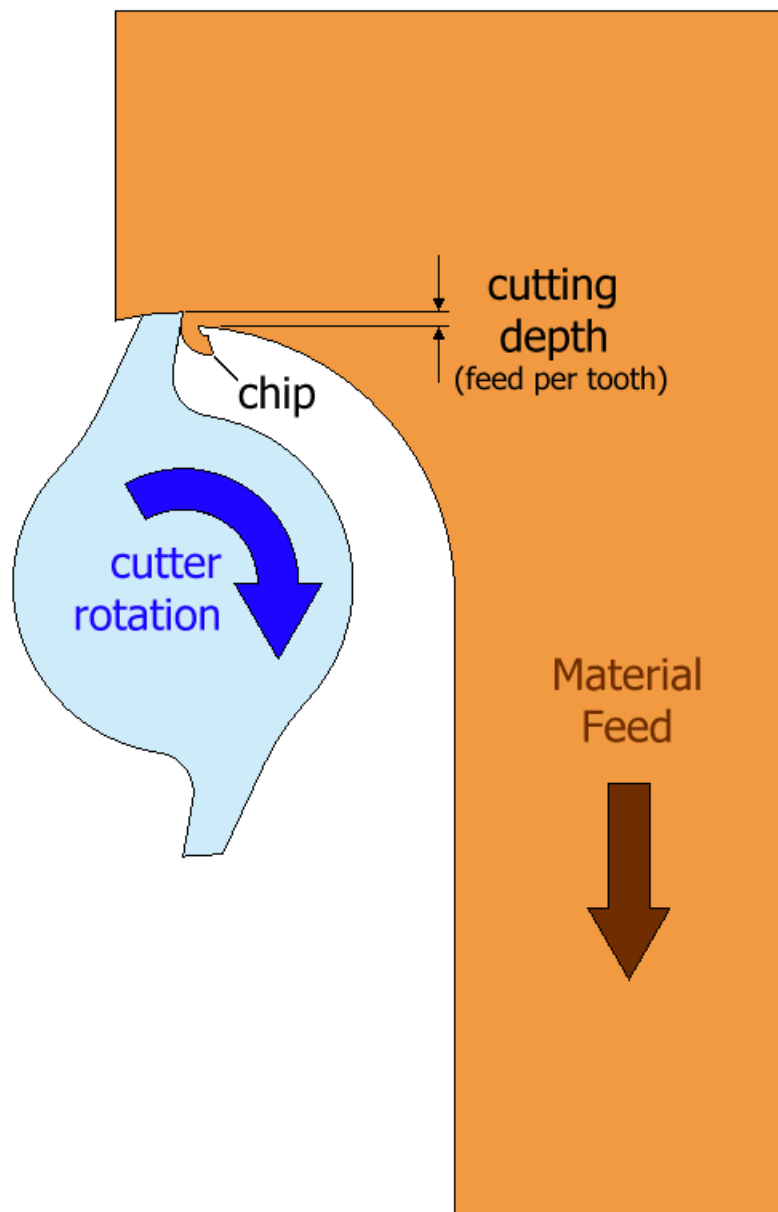
CCW

This is the setting for Counter-Clockwise direction. CCW tool paths in Router-CIM will appear in Red by default.

Climb Milling or Climb Cutting

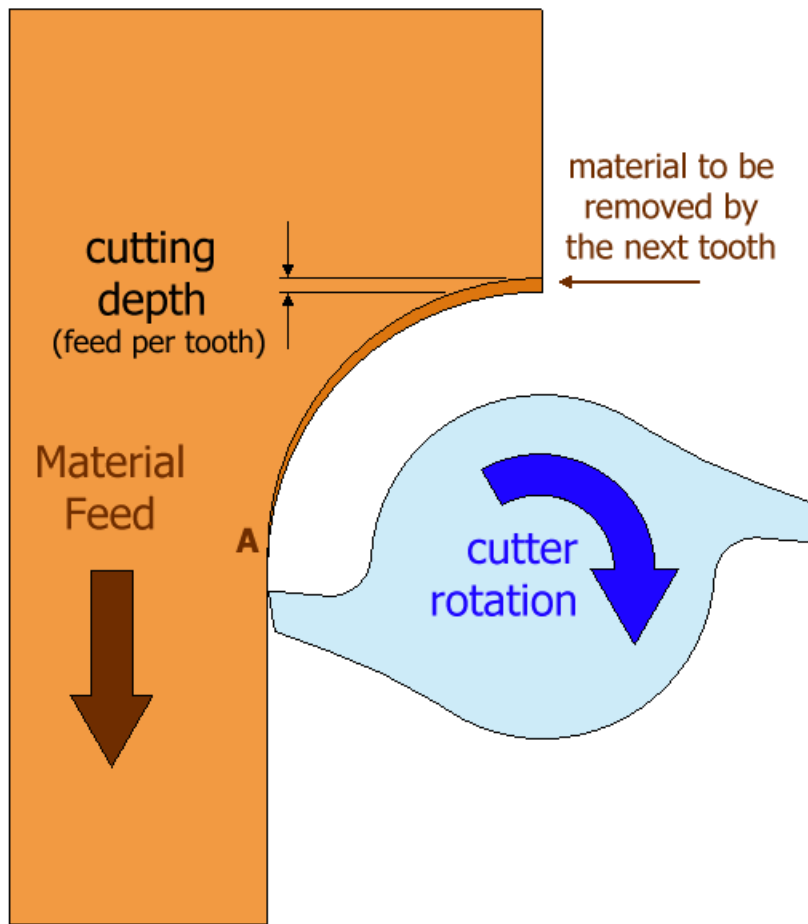
Each tooth engages the material at a definite point, and the width of the cut starts at the maximum and decreases to zero. The chips are disposed behind the cutter, leading to easier swarf removal. The tooth does not rub on the material, and so tool life may be longer. However, climb milling can apply larger loads to the machine, and so is not recommended for older milling machines, or machines which are not in good condition.

The default Cut Direction in Router-CIM is Climb Cut, both inside (CCW) and outside (CW). Change this direction to suit your needs. The drawing below displays how the direction is determined.

**Climb Cut**

Conventional Milling or Conventional Cutting

The depth of the cut starts at zero thickness, and increases up to the maximum. The cut is so light at the beginning that the tool does not cut, but slides across the surface of the material, until sufficient pressure is built up and the tooth suddenly bites and begins to cut. This deforms the material (at point A on the diagram, left), work hardening it, and dulling the tool. The sliding and biting behaviour can leave a poor finish on the material, depending on material and tool types.



Conventional Cut

Round Corners

Round Corners

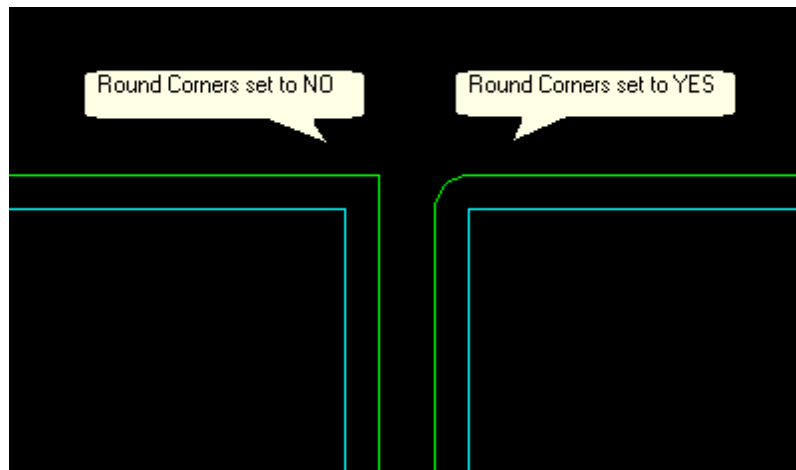
The Round Corners function, if set to Yes, will round sharp corners with a radius of 0.01. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius or the corner will be cut larger than the intended radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

N

This will set the option to NO, and no rounded corners will result.

Y

This will set the option to YES, and rounded corners will result.



Lead-In

Lead-In [v]

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits.

Only certain cycles are valid for setting the Lead-In or Lead-Out:

- Plunge-Outside Line Leads**
- Center-Line Cut**
- Right Hand/Left Hand Cut**
- T-Slot Plunge**

MULTILI

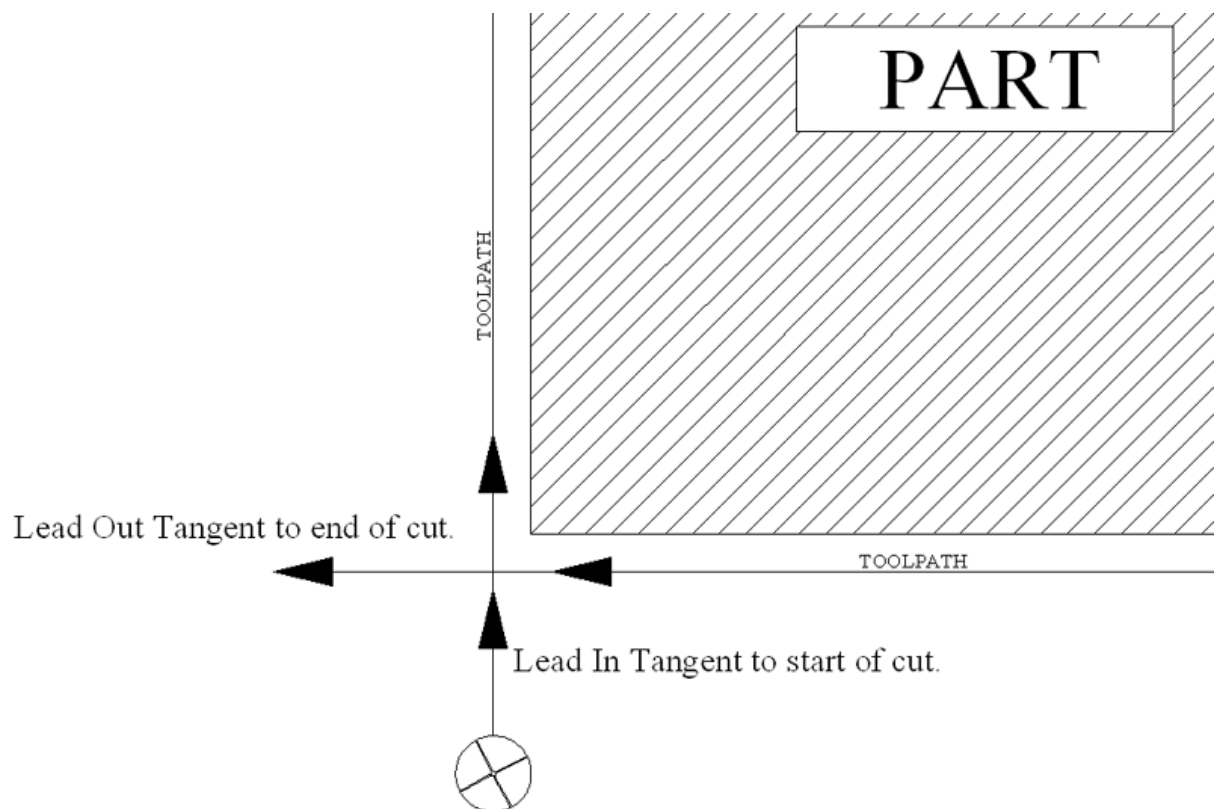
Multili is the default lead type for some cycles where there are multiple depth per pass, and Router-CIM must figure out how to move each cut down (usually ramping) between passes. This is typically not a user selectable option.

N

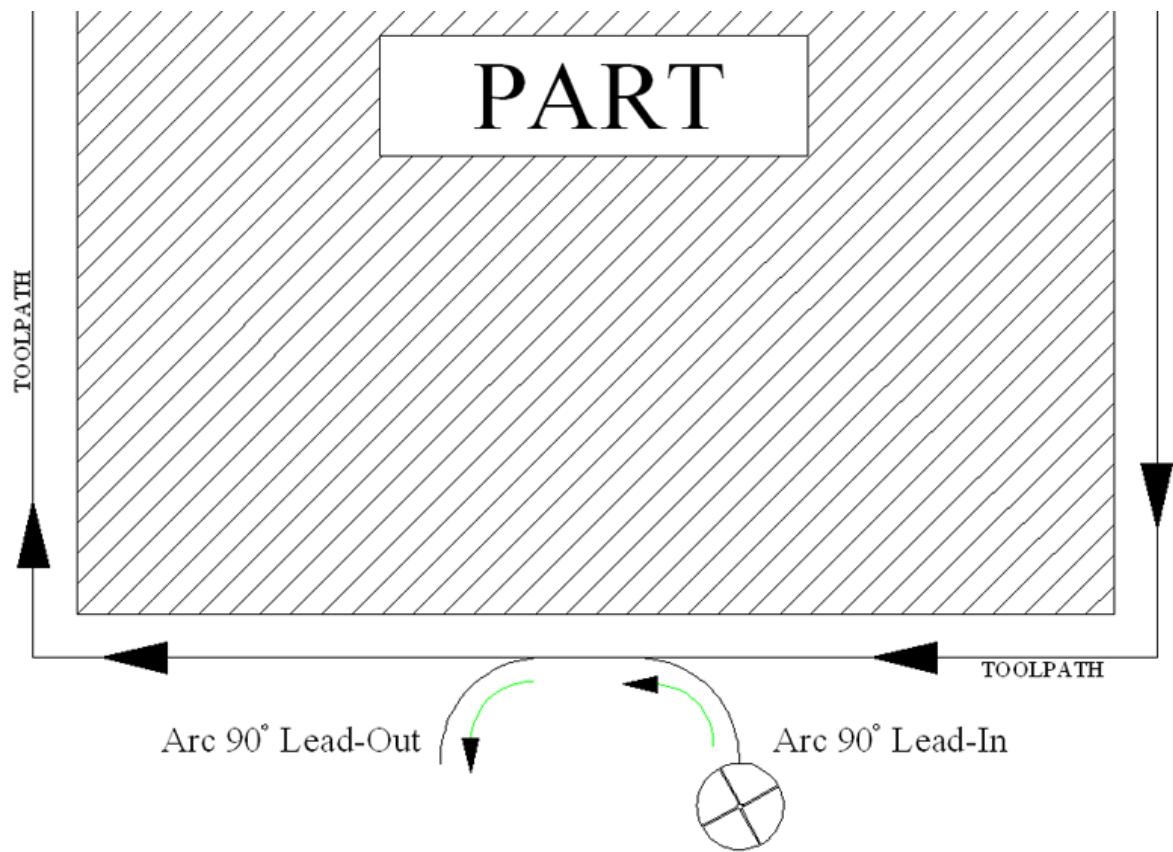
None. There will be no lead in move generated.

LNTLI

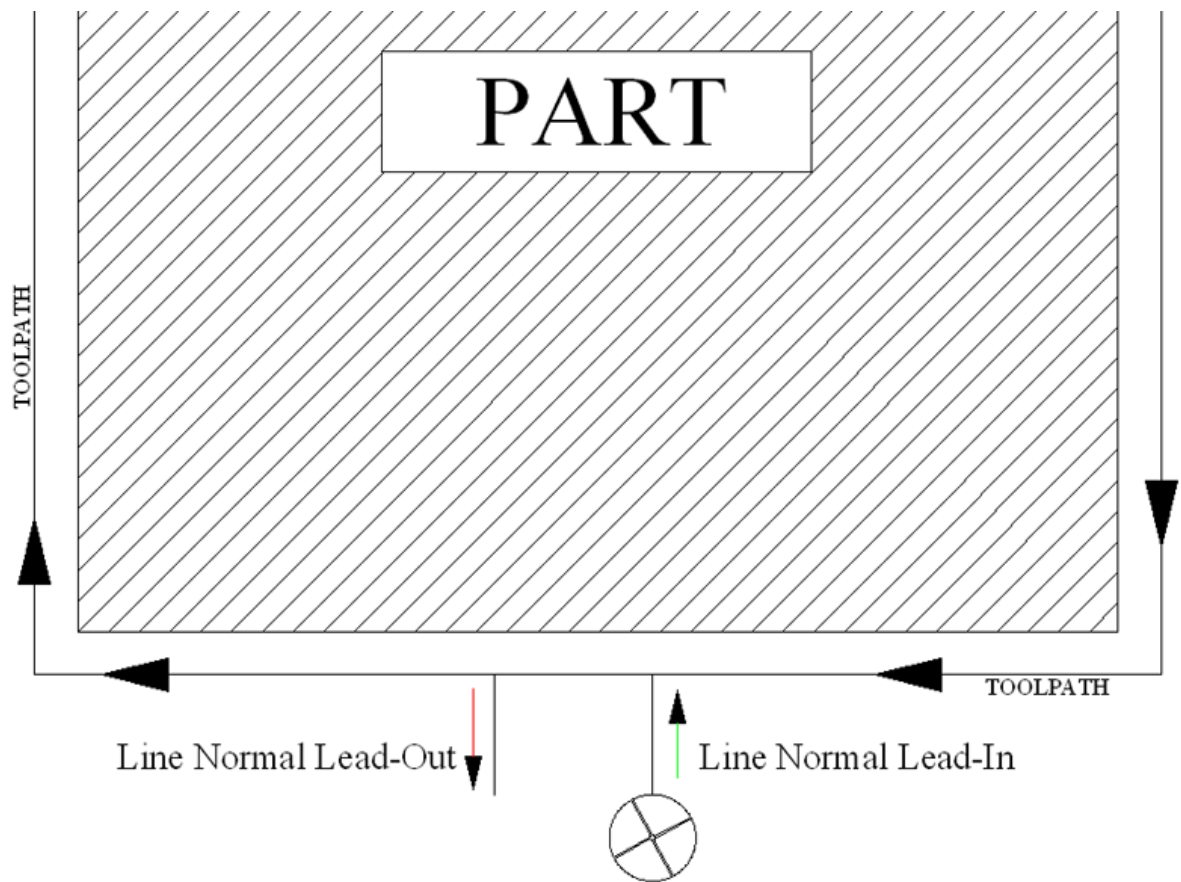
Line Tangent Lead In. This generates a move tangent to the start of the cut.

**A90LI**

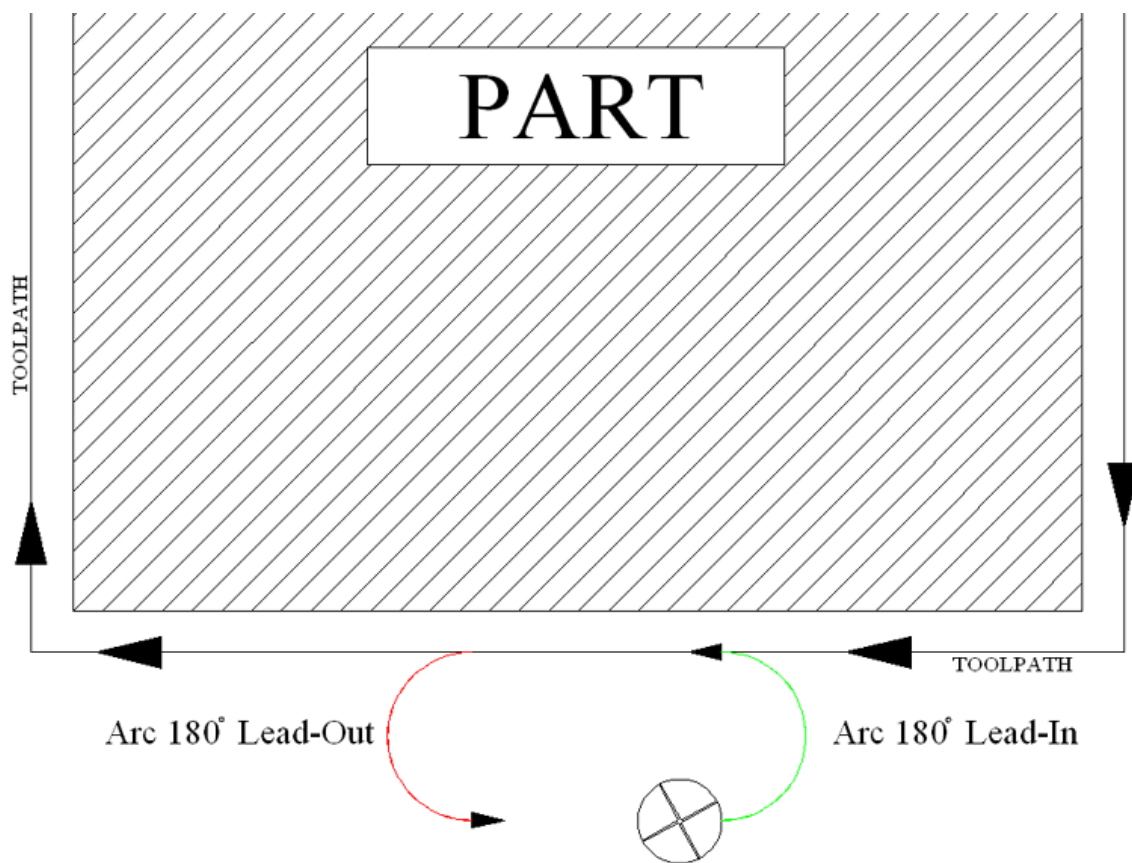
Arc 90° Lead In. Generates an arc of 90°, ending tangent to the start of the cut.

**LNNLI**

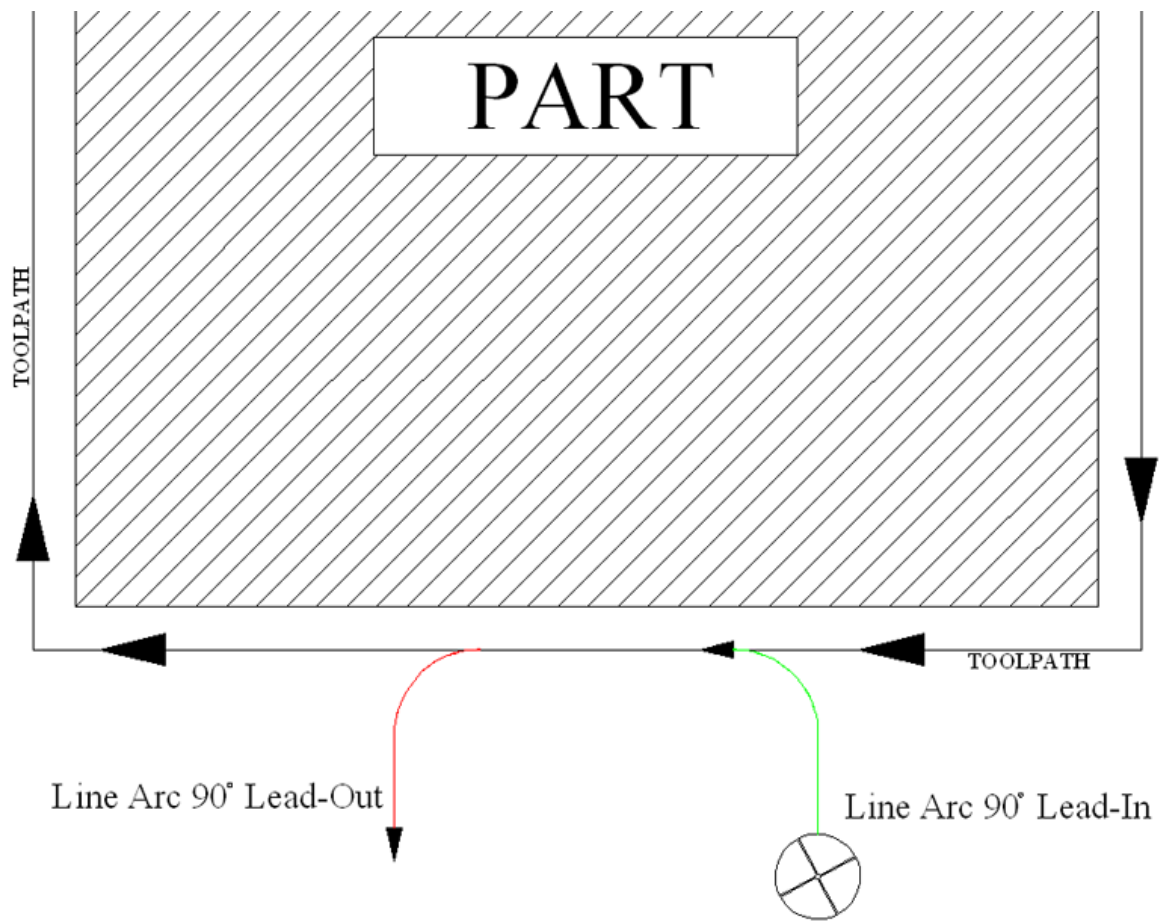
Line Normal Lead In. The Line Normal is perpendicular to the start of the cut.

**A180LI**

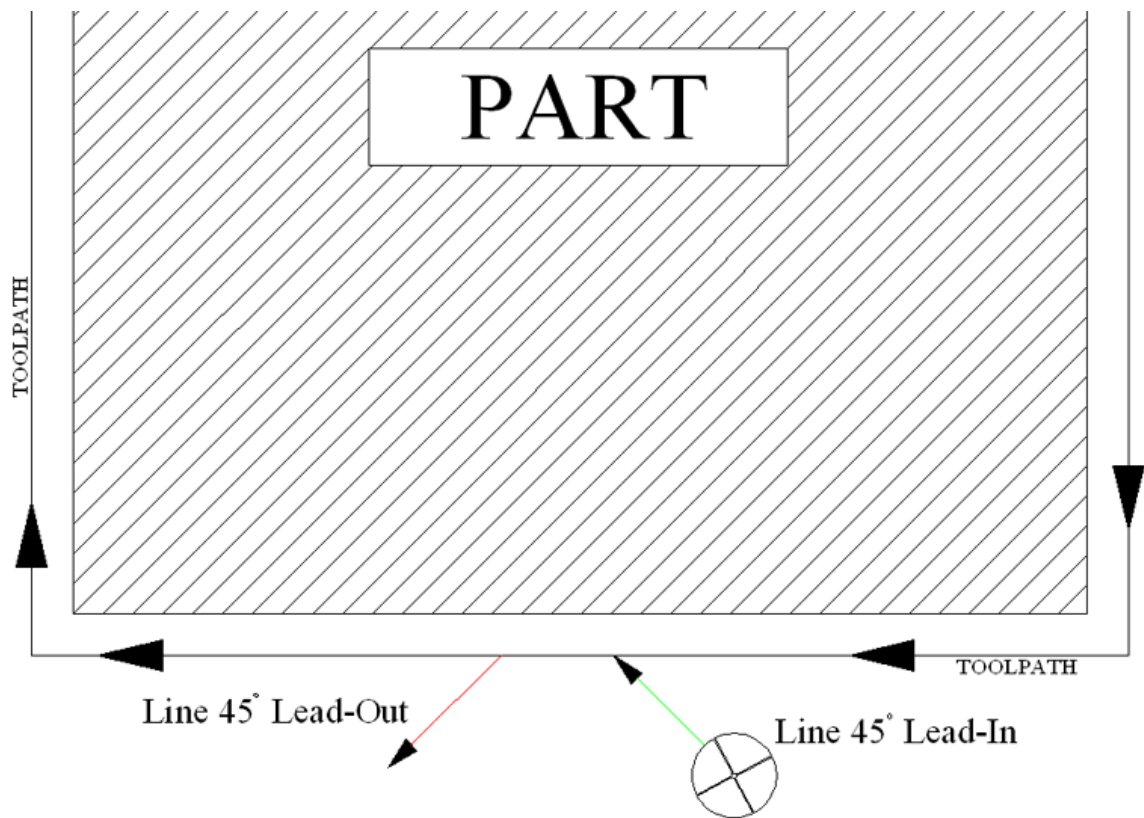
Arc 180° Lead In. An arc will be generated with an angle of 180°, ending tangent to the start of the cut.

**LA90LI**

Line and Arc 90° Lead In. There will be a short line, then a 90° arc, ending tangent to the start of the cut.

**LN-45LI**

Line 45° Lead In. This method generates a line at a 45° angle to the start of the cut.



li

This is a Router-CIM macro that will generate a line lead in, in various configurations depending on the cut settings.

This is not a user selectable option.

Creating Additional Leads

Additional Leads can be made in the /Router-CIM/Ncdwgs/(Knowledge Drawing based on the name of your postprocessor i.e. Mach1s, Mach2s, etc.) drawing by drawing a polyline on layer NC_LEADS ending at 0,0 for a lead-in. Make a block from the polyline with a 0,0 insertion point.

After the block is created you should erase the lead from the drawing and change the current layer back to Layer 0.

Save the drawing back to your /Router-CIM/Ncdwgs directory.

The next time you want to use the new lead, choose the Plunge Outside line leads cycle and enter the block name of the lead-in you chose to create, in the appropriate field in the Control Panel.

Lead-Out

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits.

Only certain cycles are valid for setting the Lead-In or Lead-Out:

Plunge-Outside Line Leads

Center-Line Cut

Right Hand/Left Hand Cut

T-Slot Plunge

MULTILO

Multilo is the default lead type for some cycles where there are multiple depth per pass, and Router-CIM must figure out how to move each cut down (usually ramping) between passes.

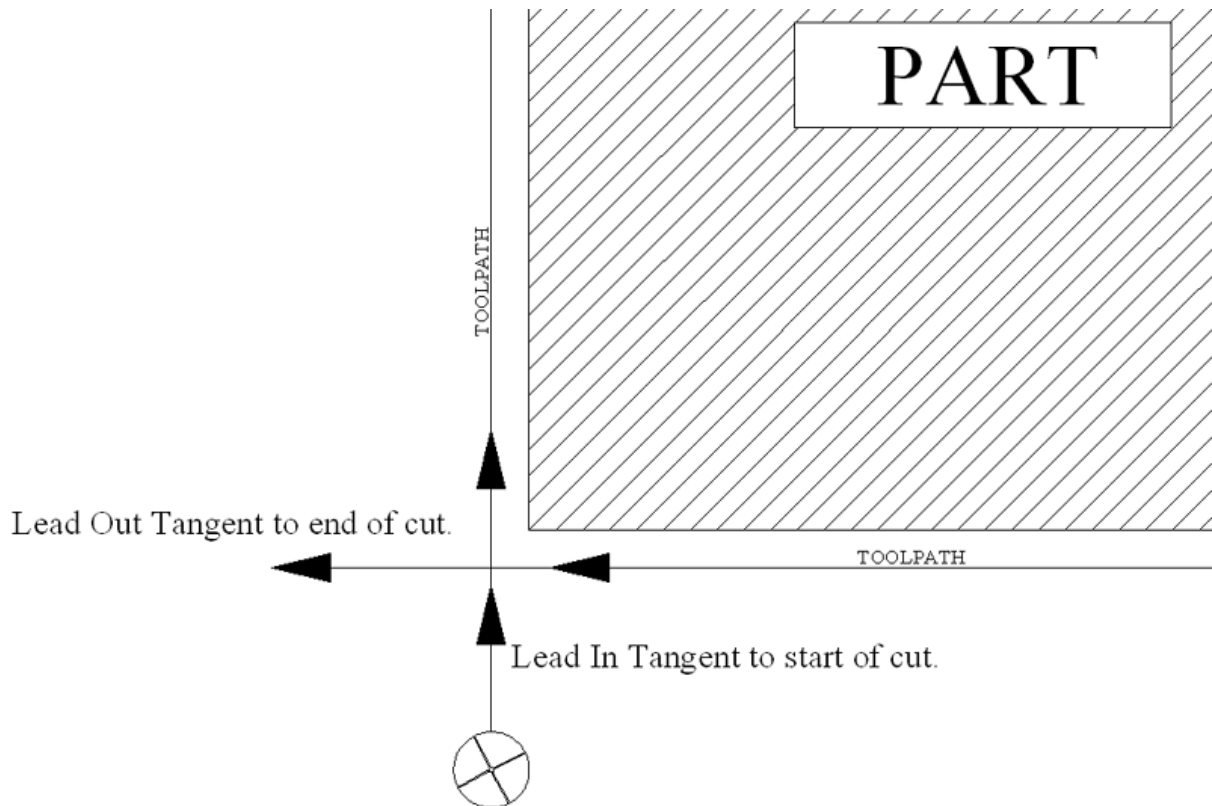
This is typically not a user selectable option.

n

None. There will be no lead out move generated.

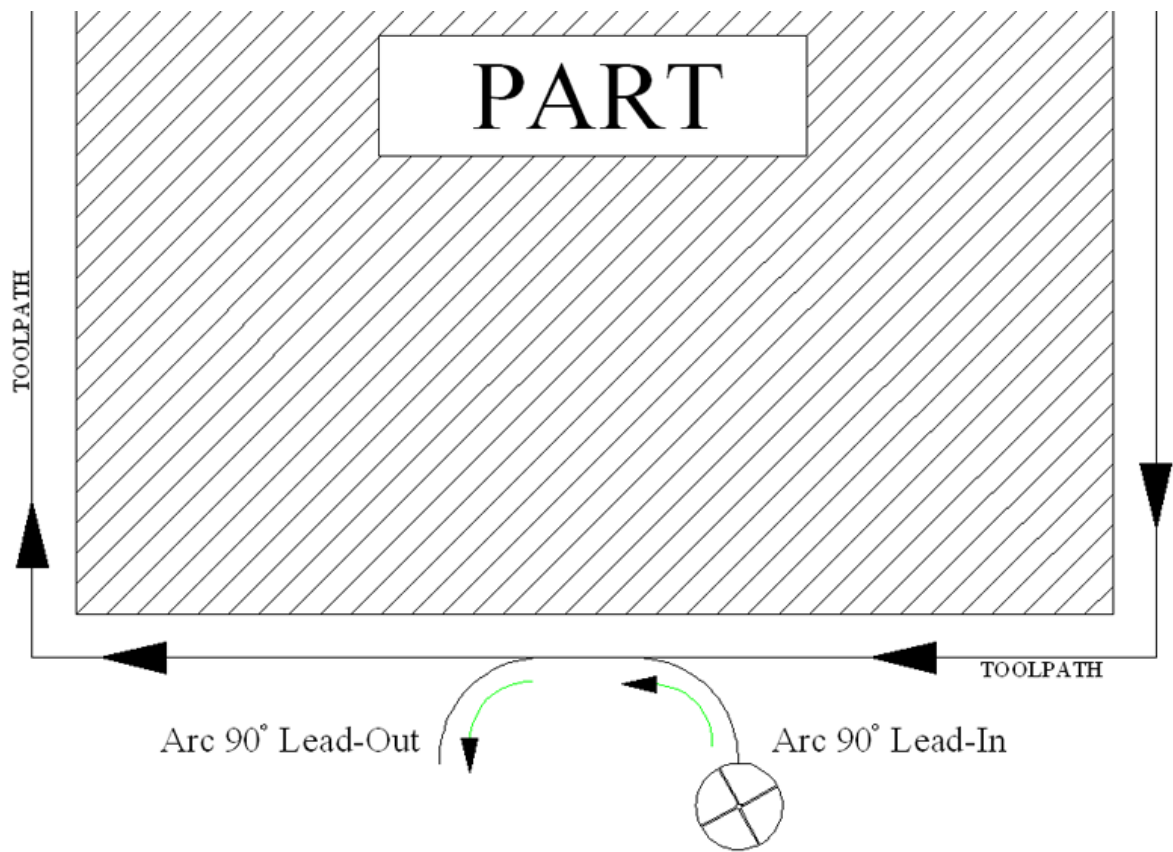
LNTLO

Line Tangent Lead Out. This generates a move tangent to the end of the cut.

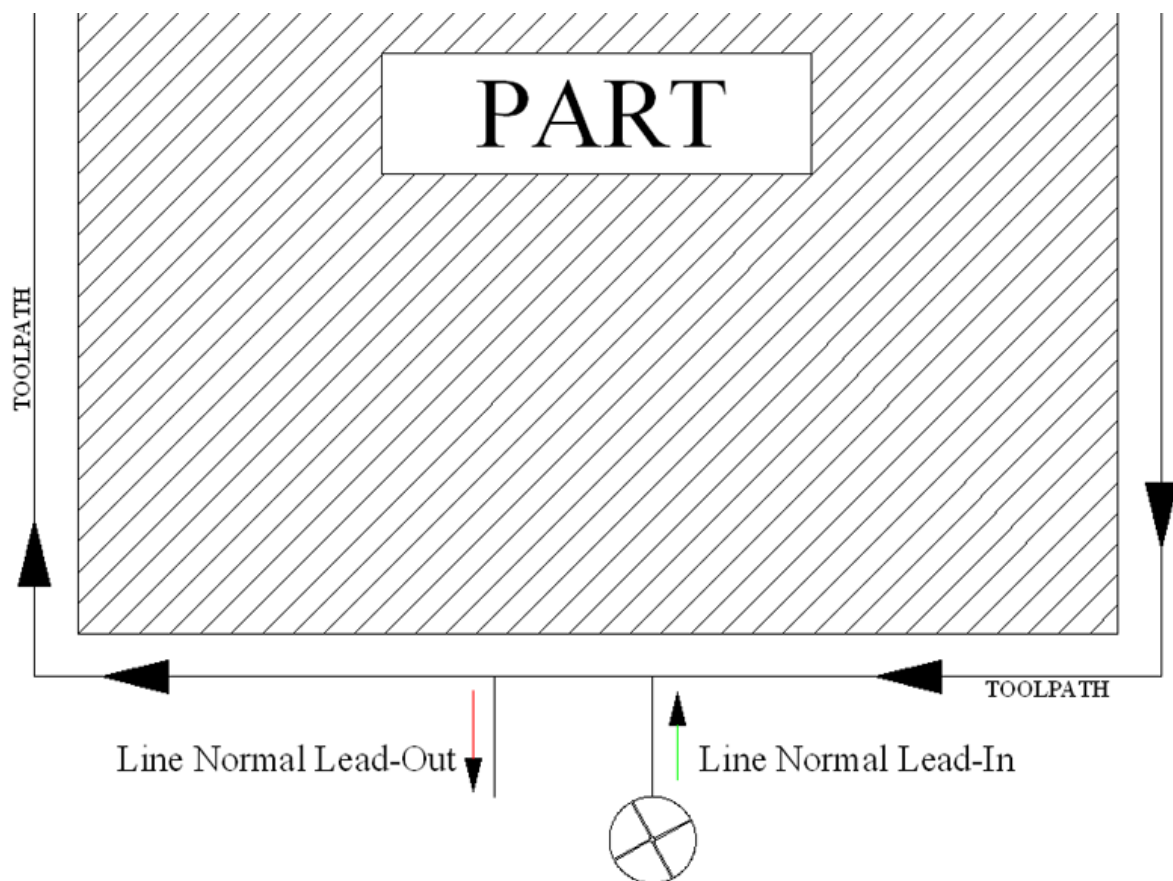


A90LO

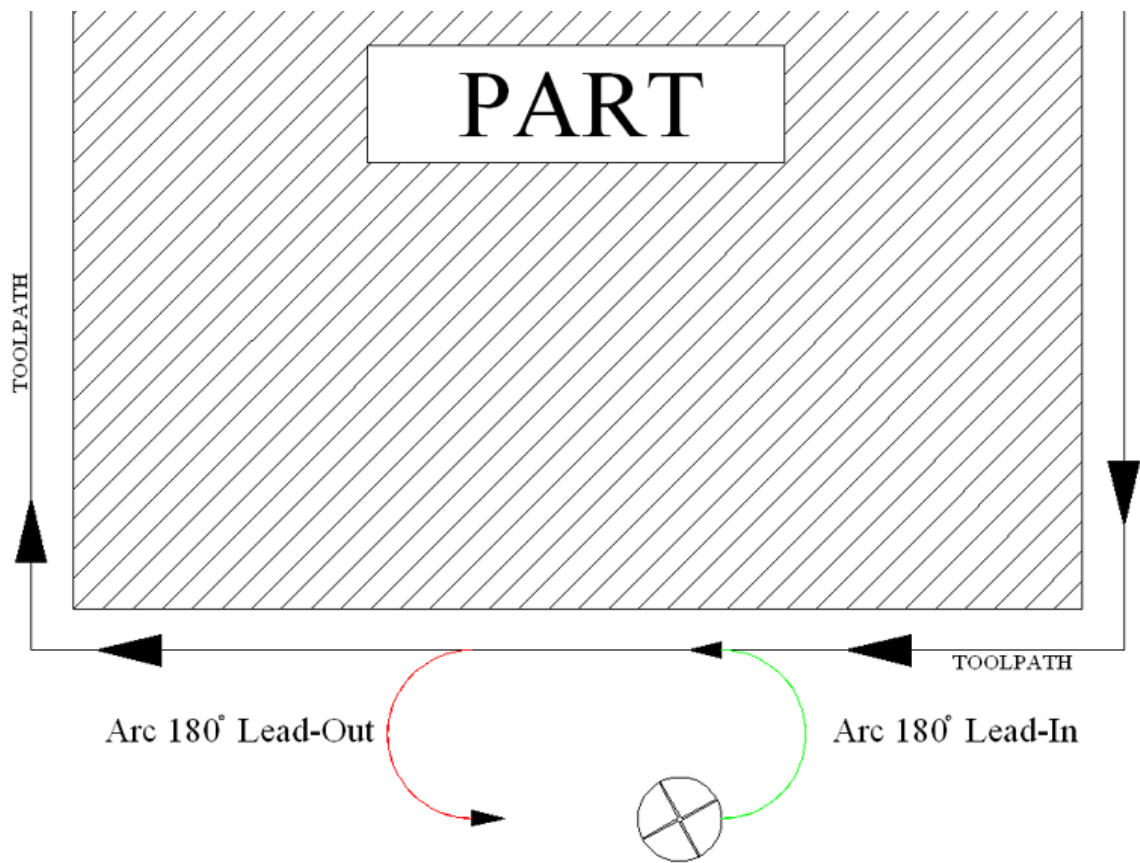
Arc 90° Lead Out. Generates an arc of 90°, starting tangent to the end of the cut.

**LNNLO**

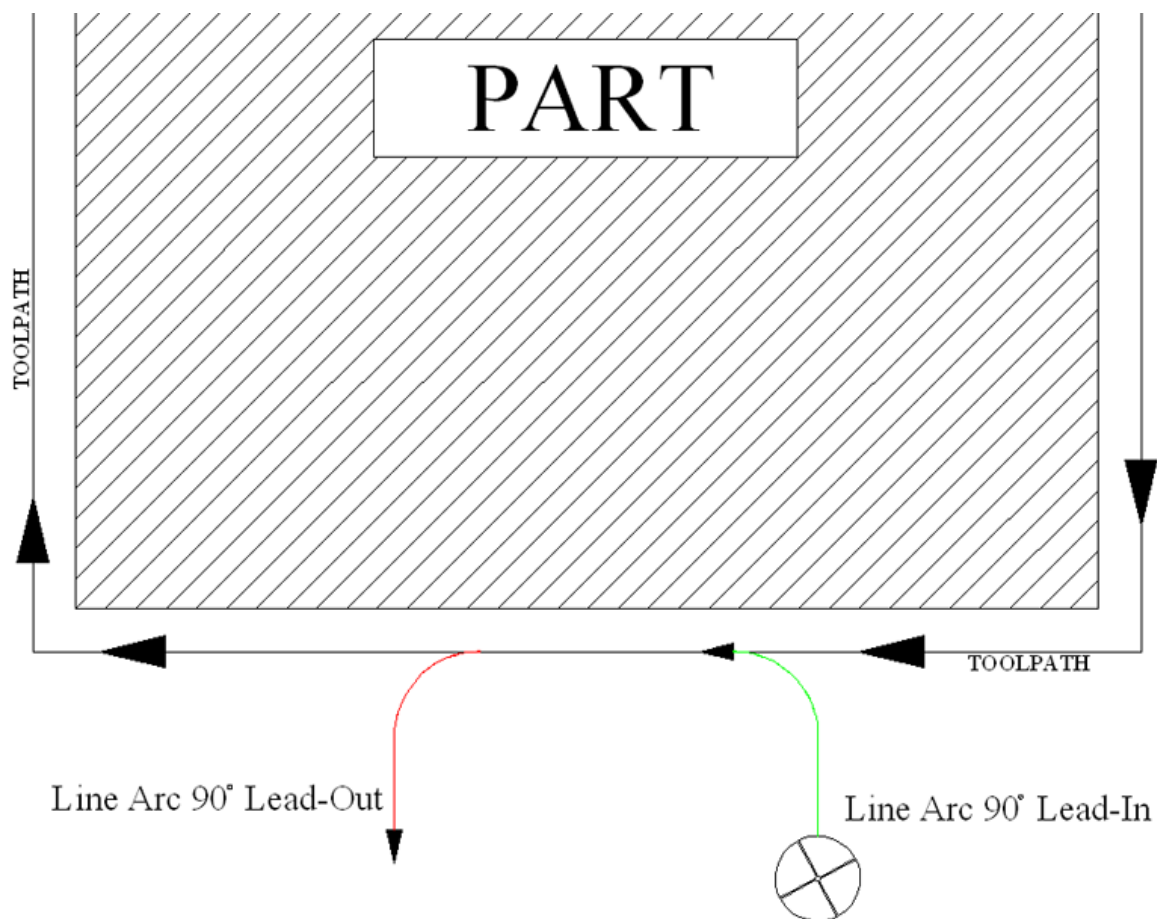
Line Normal Lead Out. The Line Normal is perpendicular to the end of the cut.

**A180LO**

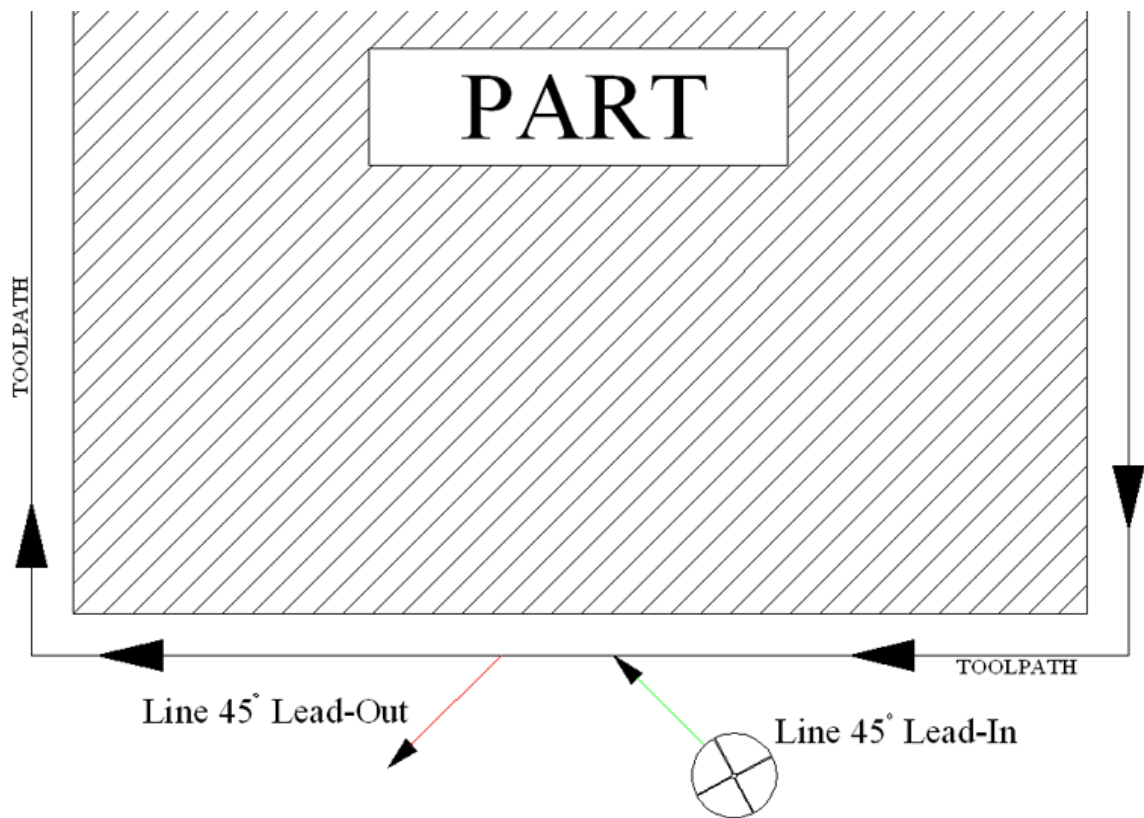
Arc 180° Lead Out. An arc will be generated with an angle of 180°, starting tangent to the end of the cut.

**LA90LO**

Line and Arc 90° Lead Out. There will be a short line, then a 90° arc, starting tangent to the end of the cut.

**LN-45LO**

Line 45° Lead Out. This method generates a line at a 45° angle to the end of the cut.



lo

This is a Router-CIM macro that will generate a line lead out, in various configurations depending on the cut settings. This is not a user selectable option.

Creating Additional Leads

Additional Leads can be made in the /Router-CIM/Ncdwgs/(Knowledge Drawing based on the name of your postprocessor i.e. Mach1s, Mach2s, etc.) drawing by drawing a polyline on layer NC_LEADS beginning at 0,0 for a lead-out. Make a block from the polyline with a 0,0 insertion point.

After the block is created you should erase the lead from the drawing and change the current layer back to Layer 0.

Save the drawing back to your /Router-CIM/Ncdwgs directory.

The next time you want to use the new lead, choose the Plunge Outside line leads cycle and enter the block name of the lead-out you chose to create, in the appropriate field in the Control Panel.

Lead Size

Lead Size

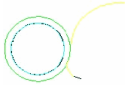
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field.

To change both leads separately, you can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

Example: Lead Size = 2.0 0.5 will make the lead-in 2.0 and the lead-out 0.5. There must be a space separating the two numbers.

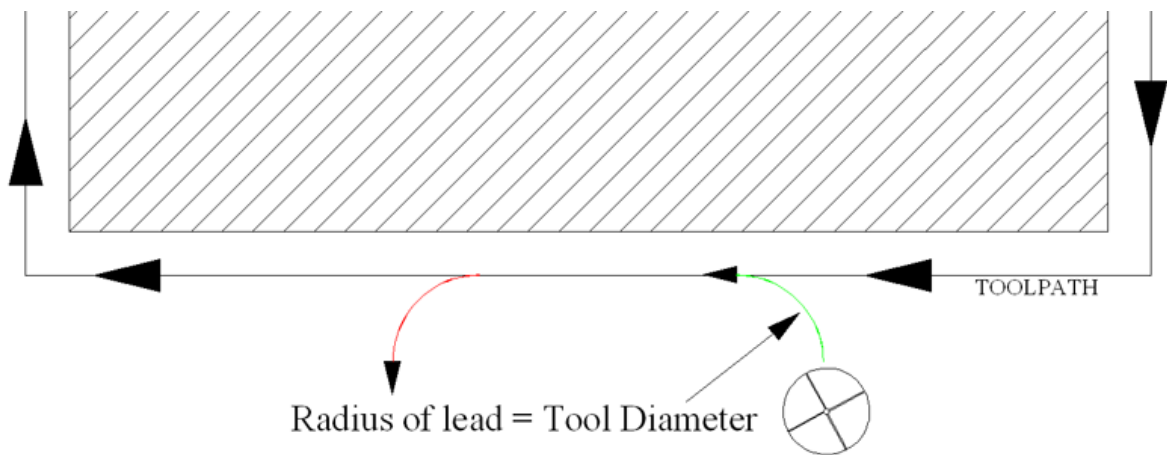
2.0 - lead-in

0.5 - lead-out



The default setting is a macro named leadscl, which will change the size of the leads according to the size of the tool.

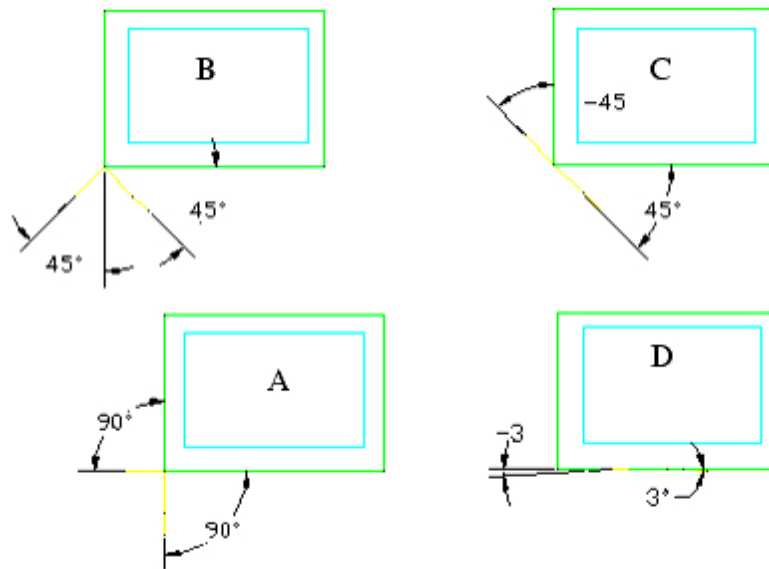
On a Plunge Outside cut for instance, the arc leads will have a size equal to the diameter of the tool.



Lead Angle

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.



The examples of lead-in and lead-out angles are:

- . •A - Plunge Outside Line Leads cycle Lead Angle is left at 0 (default), which gives lead-in and lead-out of 90°.
- . •B - Plunge Outside Line Leads cycle Lead Angle is set for 45° at lead-in and 45° at lead-out.
- . •C - Plunge Outside Line Leads cycle Lead Angle is set for 45° at lead-in and -45° at lead-out.
- . •D - Heli-Lead Outside cycle Lead Angle is set for 3° at lead-in and -3° at lead out. This would be used in a scenario in which you wanted the benefits of a Heli cycle, but you don't want the lead to bring the tool straight into the part. If the tool comes in at a slight angle it will prevent undesirable "witness marks."

Lead Feed

LeadFeed

This sets lead-in and lead-out feedrates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Note: Leadfeed parameter will only affect CNC machines using G-Code as the NC code format.

Setting the parameter to a number less than 1.0 is a percentage of max feedrate set in the Control Panel.

Setting the parameter to a number greater than 1.0 will give you an exact feedrate.

Setting the Leadfeed to 180 on a cut where the normal feedrate is 600, would allow the feedrate to be slower on the lead in than normal. Normally the lead in would be 300. In the code the lead move feedrate would appear on the same line as the lead.

```
%
:1234
N1 G00 G17 G20 G28 G40 G80 G91 Z0 M5
N2 G90
N3 G52 X0 Y0 Z0
N4 G08 P1
```

```

N5 M08
N6 (ROUTER-BIT .5 DIA.)
N7 G28 G91 Z0 M05
N8 G90 T2001 M06
N9 T102
N10 M03 S18000
N11 G00 G17 G55 X5. Y-.249
N12 G00 G43 H1 Z.25
N13 G01 X3. Z-.75 F180. <<<<<< Lead In feedrate here.
N14 X.5 F600. <<<<<< Normal cut feedrate here.
N15 G02 X-.249 Y.5 J.749

```

You could also set the Leadfeed in the example above to 0.3 and get the same result as 180 is 30% of 600.

Ramp Amount

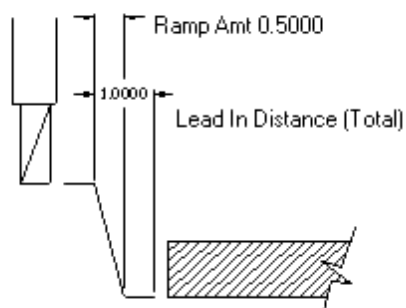
The Ramp Amount is the distance during the lead-in that the cutter spends in the ramp to the Total Depth of the Cut.

Ramp Amount

NONE

Use this feature if you want the Ramp-Outside or Ramp-Inside profiling cycle to achieve full Z depth prior to the tool being on the finished edge of the part.

For example, if you have a lead-in distance of 1.0" in 1.0" material, but want the cutter at full depth before it reaches the part, you may specify a Ramp Amount of 0.5. The cutter will ramp down to -1.0" in the first 0.5" of travel and will spend the next 0.5" of its lead-in at the maximum Cut DEPTH.



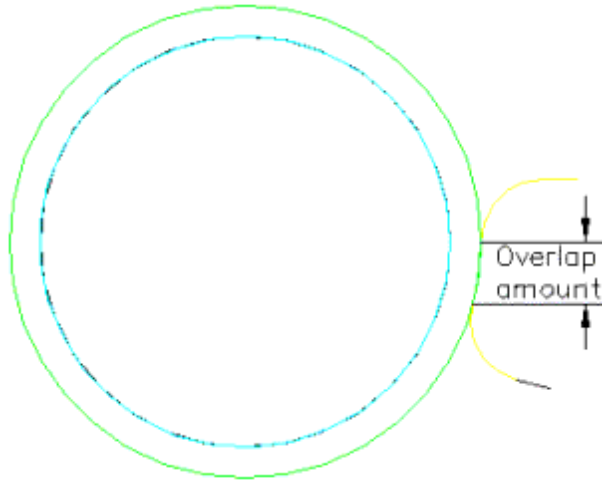
NOTE: Used with Ramp Inside and Ramp Outside cycles only.

Overlap Amt.

Overlap Amount

AUTO

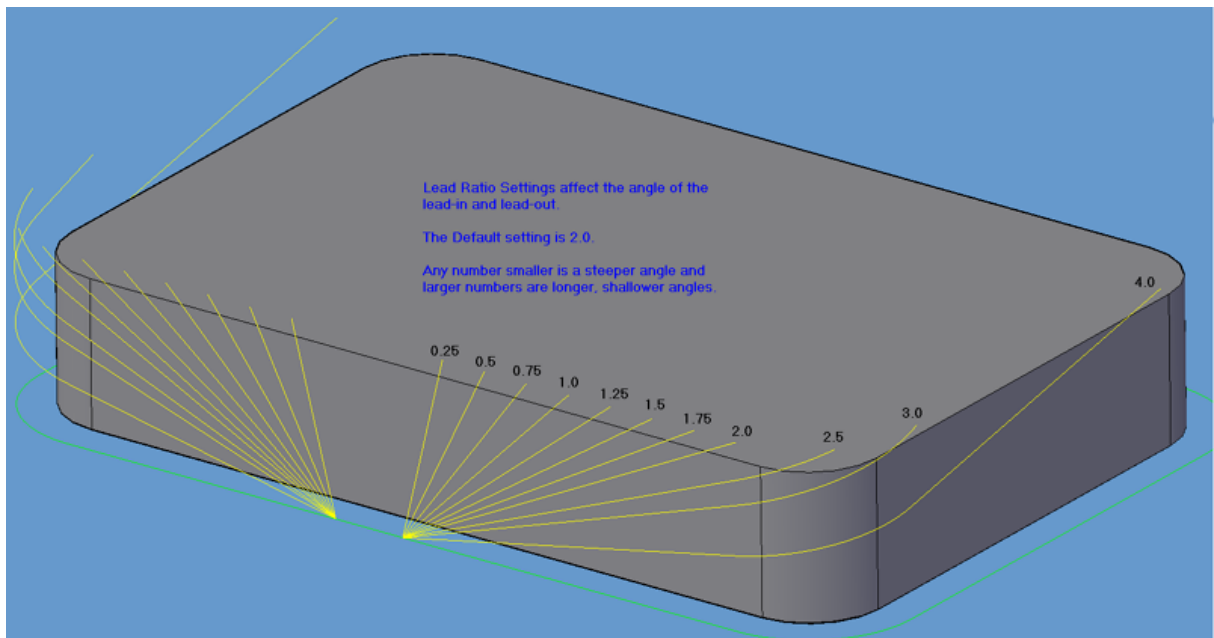
Overlap is the amount the tool travels in the cut beyond the start point before leading out of the cut. By default the Overlap amount is equal to the diameter of the tool (AUTO). You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0 in the Overlap Amt. field then the Overlap will be 1.0".



Lead Ratio

Lead Ratio:

Lead Ratio determines the angle of the ramp in Z during the lead in and lead out. You can specify the Lead Ratio as a number that reflects the percentage of the angle from its default. That means that if you want a lead that is twice the normal ramp length (shallower angle) enter 2. If you want a lead that is steeper than the default, enter .5.



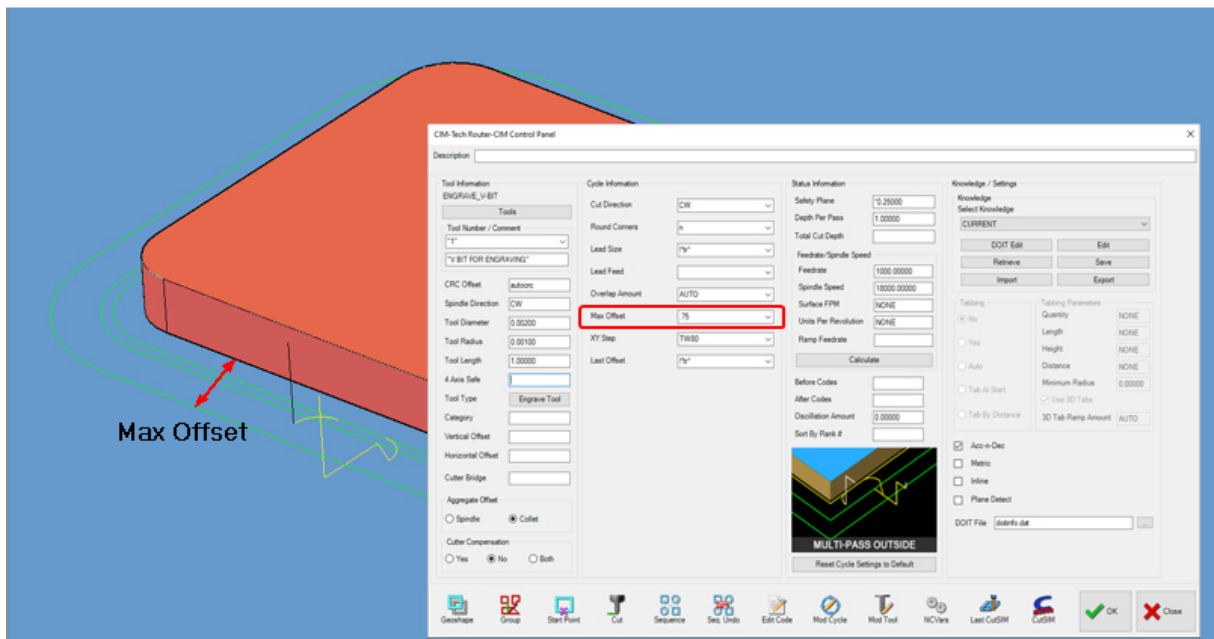
The Lead Ratio value effects both the lead in and the lead out. There is no way to have a separate angle for the lead-in and lead-out. If two numbers are input, only one is used.

MaxOffset

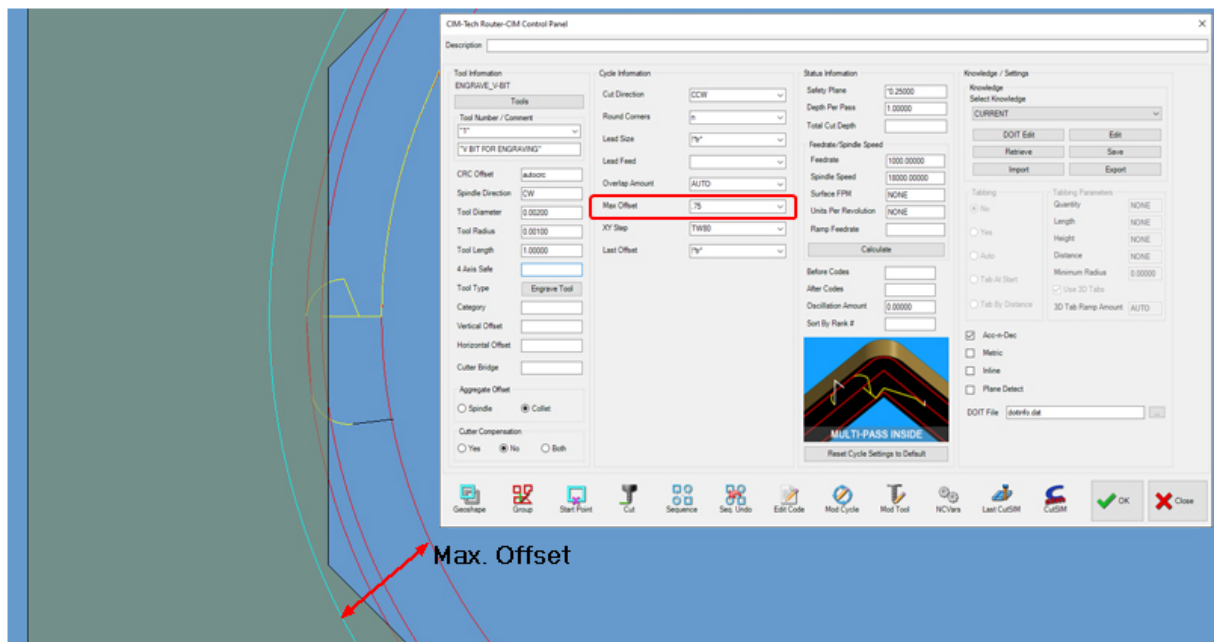
Max Offset

Max Offset is the distance from the start point to the start of the first pass on the cut. If the cut cycle is **Multi-Pass Inside**, then the Max Offset will be to the inside of the shape by the specified amount. If the cycle is **Multi-Pass Outside**, then the Max Offset will be from the start point to the outside of the shape by the Max Offset amount.

This is only the distance for the first cut. All subsequent cuts will be determined by either XY Step or Last Offset.



Max. Offset on Multi-Pass Outside tool path.

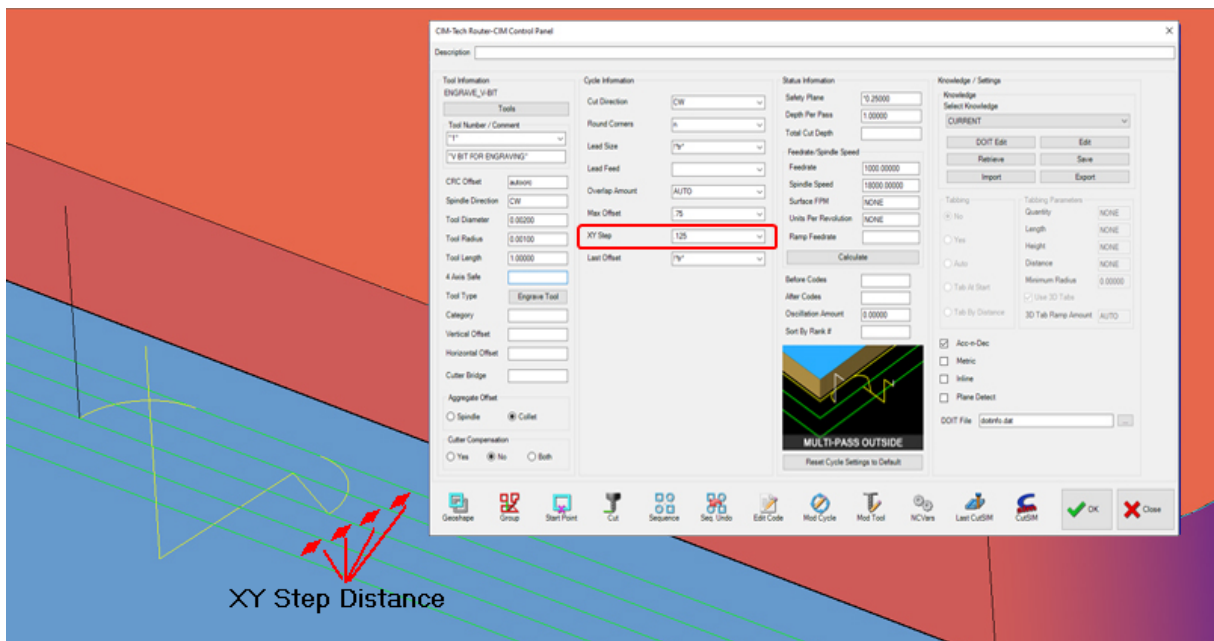


Max. Offset on Multi-Pass Inside tool path.

XYStep



The XY Step is the distance between each tool path from the first to the last (excluding the finish pass) on the **Multi-Pass Inside** and **Multi-Pass Outside** cycles.



You can specify the distance as a numeric value, or you can make a task to calculate a step over amount and place the task name in the XY Step field. An example of a task is TW80, which looks at the tool diameter field and then places an amount equal to 80% of that value in this location.

The task must be in proper lisp-task form and placed in a {taskname}.tsk file in the ncprog\ncsources folder. For example, make a file called TW60.tsk with Notepad and place it in the C:\Router-CIM\Ncprog\Ncsources folder. Place the following code inside the file.

```
(* *TW* 0.6)
```

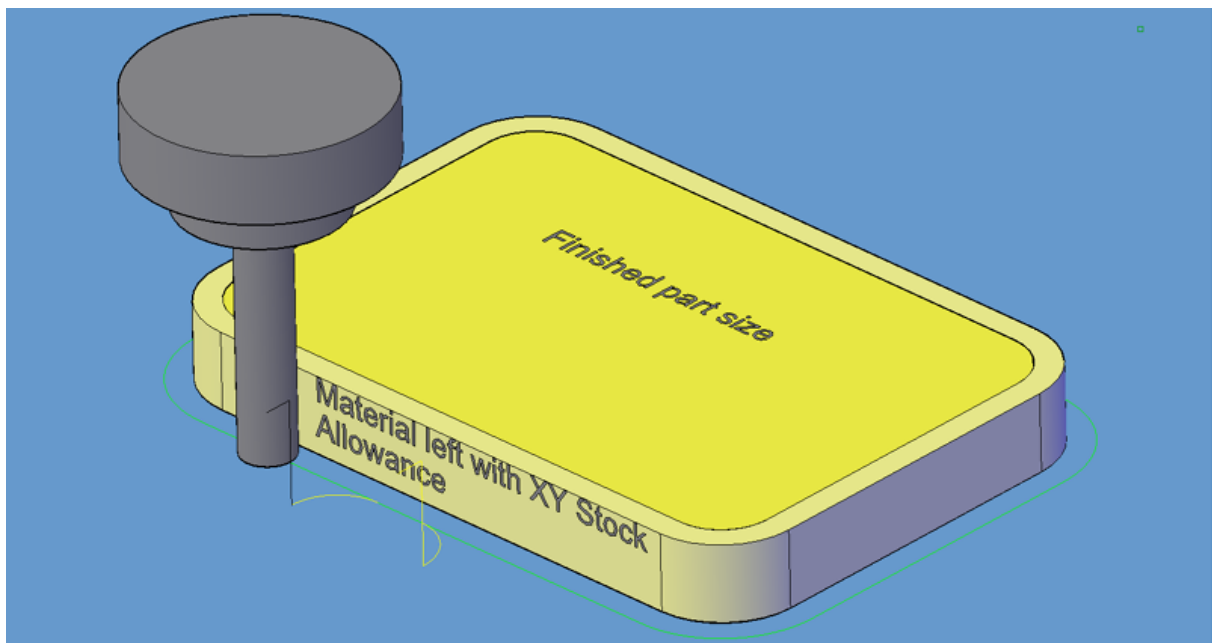
Save the file and then place the syntax TW60 in the XY Step field after starting Router-CIM again. The step over will then be 60% of the tool diameter. The code reads as follows:

(multiply ToolDiameter by 0.6) so (* 0.5 0.6) would be 0.3. So a step over of 0.3 would be set for the spacing for any pass that is not the finish pass.

XY Stock Allowance

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.65 x 4.25, by adding .125 to the offset of the tool path all the way around the part.



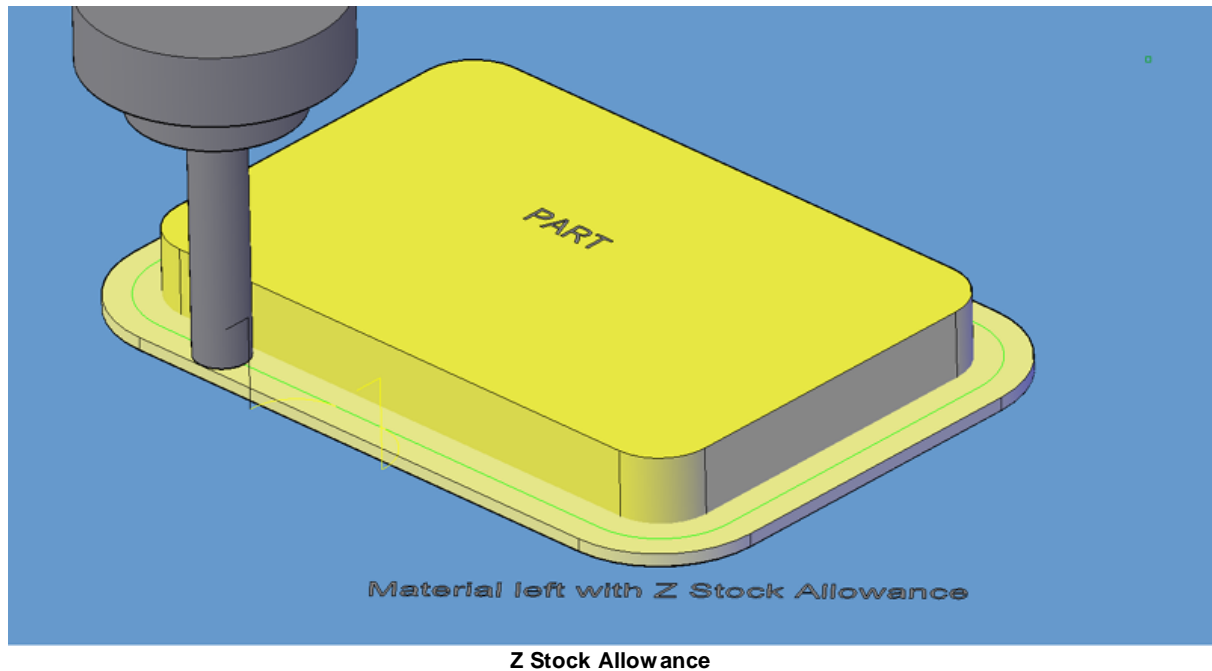
XY Stock Allowance.

Z Stock Allowance

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

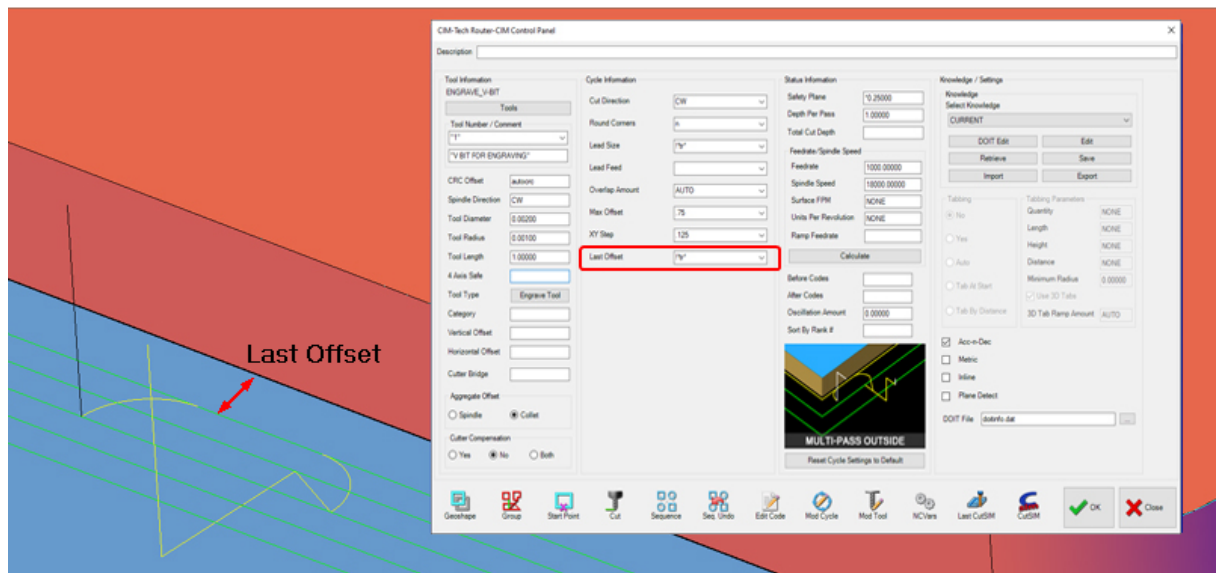
Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.



Last Offset

Last Offset

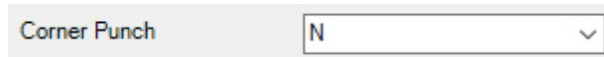
The Last Offset is the distance from the finish pass to the edge of the shape being cut. This can be any numeric value, but if it exceeds the XY Step value, the cutter will not contact the part on the finish pass. Typically this is set to the radius of the tool in a rough-cut, finish-cut scenario. You can leave some material on the part if you want to clean up the part with a finish cutter.



Last Offset distance on Multi-Pass Outside tool path.

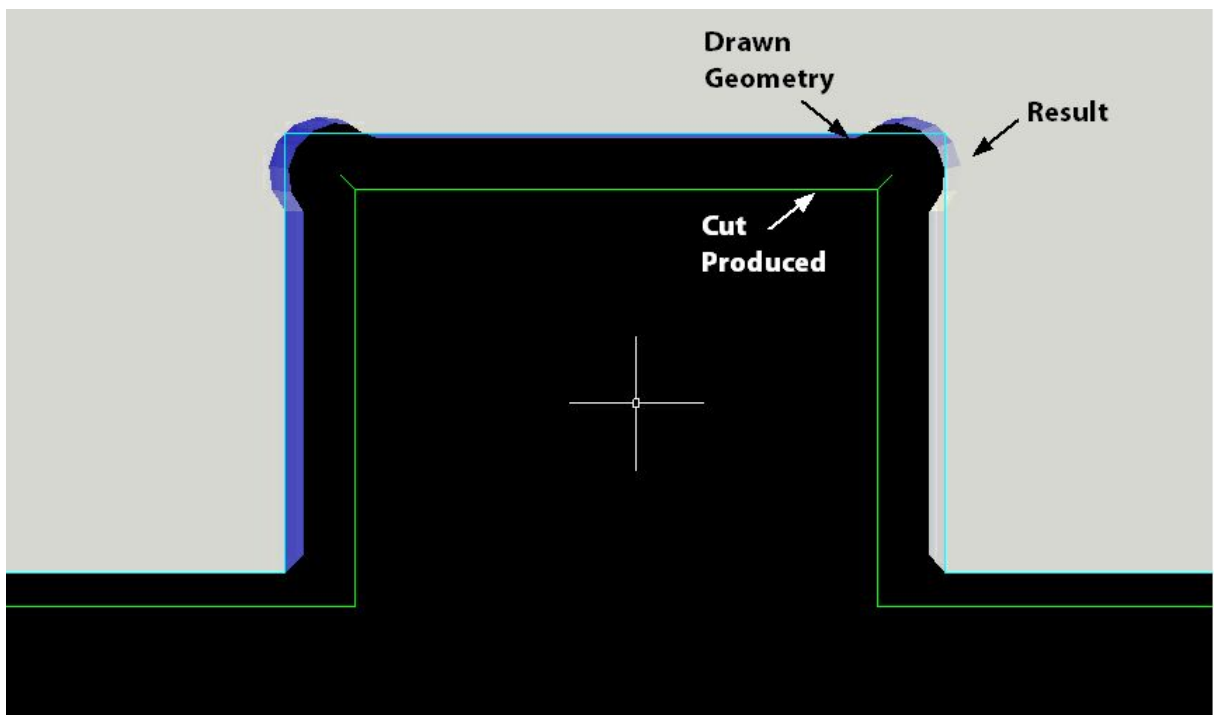
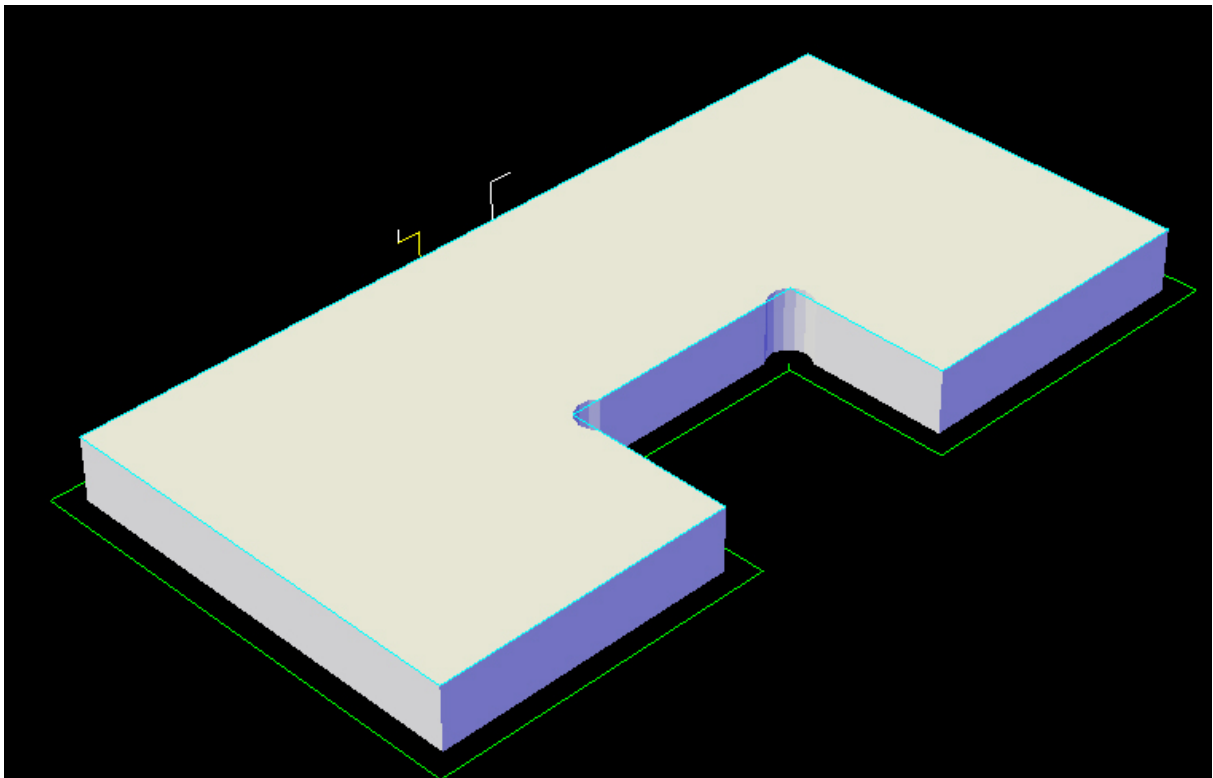
Cornerpunch

The Cornerpunch feature will find any interior angle (less than 180 degrees) and extend the cut path into the angle by required amount based on the tool size in order for a piece to fit flush within the angle without having to draw the geometry.



Acceptable options are N and Y.

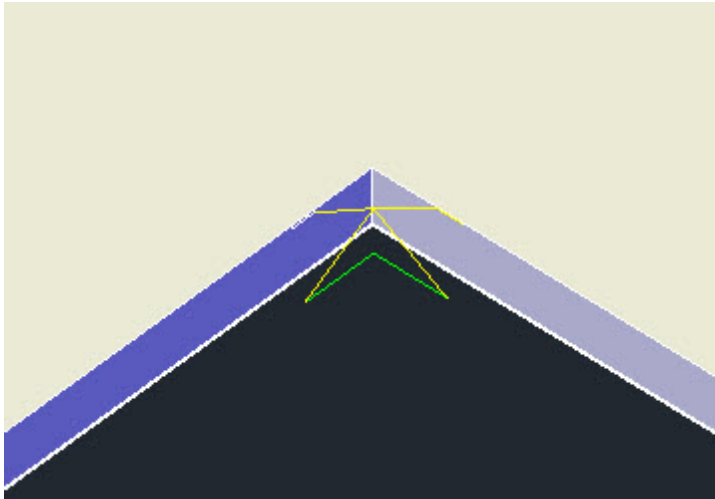
This feature will work on OUTSIDE and INSIDE single pass profile cutting cycles when set to Y.



Note: The Cornerpunch feature only works with Cutter Compensation set to NO or BOTH.

Corner Clean

The Corner Clean feature will find any angle and draw a piece of geometry for a tool to follow in order to use the specified tool to clean or tighten a radius on an angle. This is a common practice when using a larger tool to do the initial cut for speed and be able to come back with a smaller radius tool in only the angled corners to tighten the radius. The Corner Clean feature works on closed geometry on either an inside or outside feature.



This feature will work on OUTSIDE and INSIDE single pass Center-line Cut or Center-line Ramp cutting cycles.

To use the Corner Clean feature, you will start by creating a knowledge with the correct tool and either Center-Line Cut or Center-Line Ramp cycle.

Note: When selecting Center-Line Cut or Center-Line Ramp, you will need to make sure to set the 'Offset Dim' to OFFSZ and 'Cut Side' to either OUTSIDE or INSIDE depending on the the type of geometry you are going to be assigning the knowledge to in order to get the correct offset for cutter compensation.

Cycle Information	
Offset Dim	OFFSZ
Cut Side	OUTSIDE

Before saving the knowledge, select 'OK' on the control panel. When in AutoCAD, type in ST and hit enter.

This will bring up the Edit Parameters dialog box for the Status variables. Scroll down till you see line [12]Aux. POST Command and highlight that line:

Edit Parameters

Parametric name: STATUS

Icon ☐

Description	Value
[10]Positioning Distance....	
[11]Aux. POST Command.....	
[12]Aux. POST Command.....	
[13]Aux. POST Command.....	
[14]Aux. POST Command.....	
[15]Aux. POST Command.....	
[16]Aux. POST Command.....	(nil nil nil nil nil nil nil nil nil nil 0.0 nil nil "CEN
[17]Aux. POST Command.....	
[18]Aux. POST Command.....	

Edit Value of Selected Parameter

Aux. POST Command.....

OK Cancel Notes.. TaskInfo... Write...

Permanent Editing

In the 'Edit Value of Selected Parameter' area, select in the Aux. Post Command dialog window and type in cornerclean followed by a space and then the distance you want the corner clean feature to be.

Edit Parameters

Parametric name: STATUS

Icon ☐

Description	Value
[10]Positioning Distance....	
[11]Aux. POST Command.....	
[12]Aux. POST Command.....	
[13]Aux. POST Command.....	
[14]Aux. POST Command.....	
[15]Aux. POST Command.....	
[16]Aux. POST Command.....	(nil nil nil nil nil nil nil nil nil nil 0.0 nil nil "CEN
[17]Aux. POST Command.....	
[18]Aux. POST Command.....	

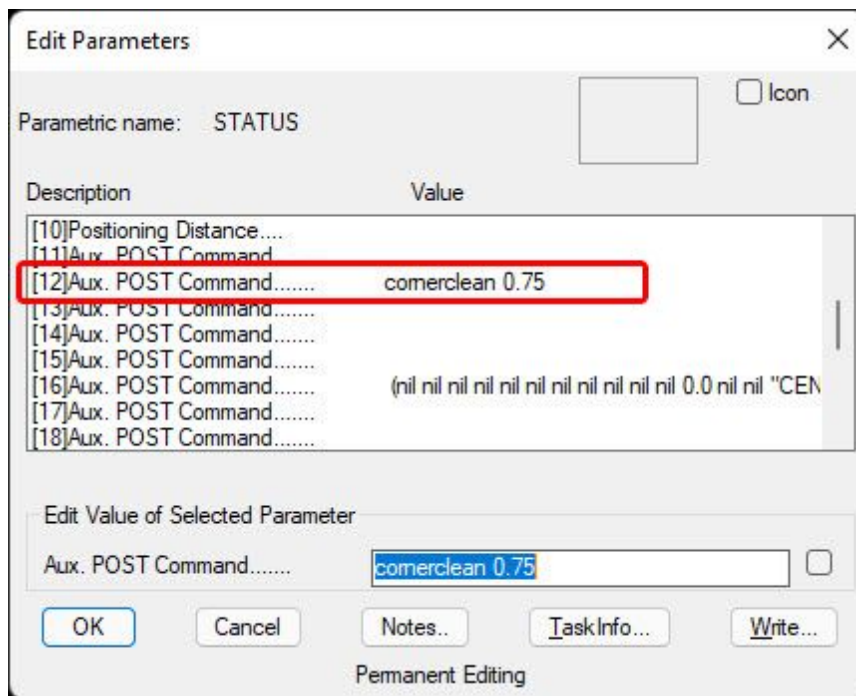
Edit Value of Selected Parameter

Aux. POST Command..... cornerclean 0.75

OK Cancel Notes.. TaskInfo... Write...

Permanent Editing

Hit the enter key to update line [12]:



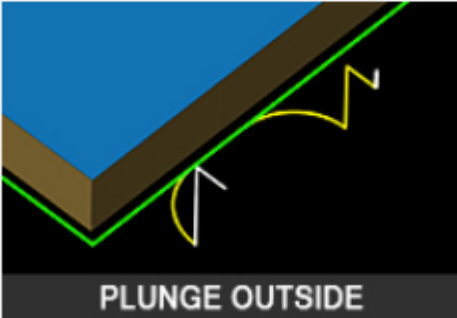
Line [12] should now have the updated information.

Go back to the Router-CIM Control Panel and select the 'Save' button to save the knowledge. Once you have the knowledge saved, you can then go to the DOIT file and assign the knowledge to the correct layer that you would like to use the Corner Clean feature on.

Note: Once the knowledge using the Corner Clean feature has been saved, if creating another knowledge that you do not intend on using the Corner Clean feature, first select a different saved knowledge from the control panel that does not use the Corner Clean feature prior to creating the next knowledge.

Status Information

Status Information is data related to the feed rates, spindles speeds, depths of cut etc. for the type of material and cutters available. The field has several parameters and each is explained in the following sections.

Status Information	
Safety Plane	<input type="text" value="*0.25000"/>
Depth Per Pass	<input type="text" value="1.00000"/>
Total Cut Depth	<input type="text"/>
Feedrate/Spindle Speed	
Feedrate	<input type="text" value="1000.00000"/>
Spindle Speed	<input type="text" value="18000.00000"/>
Surface FPM	<input type="text" value="NONE"/>
Units Per Revolution	<input type="text" value="NONE"/>
Ramp Feedrate (%)	<input type="text" value="100.00000"/>
<input type="button" value="Calculate"/>	
Before Codes	<input type="text"/>
After Codes	<input type="text"/>
Oscillation Amount	<input type="text" value="0.00000"/>
Sort By Rank #	<input type="text"/>
	
<input type="button" value="Reset Cycle Settings to Default"/>	
<input checked="" type="checkbox"/> Acc-n-Dec	
<input type="checkbox"/> Metric	
<input type="checkbox"/> Inline	
<input type="checkbox"/> Plane Detect	
DOIT File	<input type="text" value="doitinfo.dat"/> <input data-bbox="1372 1203 1421 1234" type="button" value="..."/>

Safety Plane

Safety Plane	<input type="text"/>
--------------	----------------------

This field is the Z axis SAFETY PLANE to which the tool retracts to between CUTS, or after a cutting cycle. The code generated will have a Z move to this location in rapid, then a Z move to the feed distance to material (parameter #29 in Modify Tool page), and the next Z axis move will have a feed rate attached to it. After CUTTING, the code will generate a move back to this location and then proceed either to the next Cut or to the next tool, etc.

NOTE: Placing an asterisk (*) in this field before the value will ensure that the move is absolute, instead of incremental. For instance using *0.25 will return the cutter to .25 above Z0, however, using 0.25 (no *) will return the cutter to a point .25 units above the last Z position only.

Depth Per Pass

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to cut through the part, you could set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0. In the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

Total Cut Depth

The Total Cut Depth is the final depth, in Z, you want the tool to cut down to based on the top of the geometry, regardless of the number of passes made. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to cut into the material. If the Depth Per Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

There are 5 options for Total Cut Depth:

- 1) You can enter a number into this field. This will instruct Router-CIM to always go down to the set amount you have defined.
- 2) You can leave the field blank which will instruct Router-CIM to obtain the value by the NEGATIVE thickness that has been assigned to the geometry.

NOTE: Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (/ -0.01 for example) in this field. Router-CIM will take the negative part thickness (-0.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -0.76. If you would like to leave material, the forward slash can be followed by a positive number (/0.03 for example). With the same negative part thickness mentioned earlier (-0.75 for example), the Total Cut Depth would be -0.72

- 3) If you are running these parts through Router-CIM Automation Suite, you can use a **capital M** which will instruct Router-CIM to obtain the value by the thickness of the material that you have assigned the part to in Router-CIM Automation Suite.

NOTE: When you have Total Cut Depth set to material thickness, you can use a forward slash (/) followed by a negative value (M/-0.01 for example) in this field. Router-CIM will take the material thickness that the part was assigned to in Router-CIM Automation Suite (0.75 MDF for example), and the negative value following the M slash and calculate the Total Cut Depth. In this case the part would be cut to -0.76. If you would like to leave material, the forward slash

can be followed by a positive number (M/0.03 for example). With the same material thickness mentioned earlier (0.75 MDF for example), the Total Cut Depth would be -0.72.

4) You can set the field to a **capital P followed by a forward slash (/) and a number from 0 to 1**. The capital P stands for percentage. The cut depth will go down the percentage of the NEGATIVE thickness that has been assigned to the geometry.

For example, if the field was set to P/.6 when used to cut a piece of geometry that was drawn with a NEGATIVE thickness of -1, the resulting cut would go down 0.6 or 60%.

5) You can set the field to a **capital T followed by a forward slash (/) and a task name**. The capital T stands for TASK NAME. The cut will follow the task routine that has been called.

For example, if the field was set to T/SPIRAL when used to cut a piece of geometry, the cut will follow the task definition which will be used to get the total cut depth definition.

The following section on [Material Thickness Compensation](#) can directly affect Total Cut Depth.

Material Thickness Compensation

If you program Z0 off the top of the spoil board instead of using the top of your part, you still program as if the top of part is Z0. To set Router-CIM for spoil board as Z0, put in your material thickness in the Material Thickness Compensation field shown below. The interface below is arrived at after a cut is made and you are ready to Sequence.

From the Sequence screen you choose the Options button to the right of "Make NC Code."

NC SEQUENCE BUILDER

Sequence: NCP_SEQ_1 Cut Block Sequence **Clear All**

Sequence Definition

☒ Develop a Cut Sequence

☐ Sort Cuts by: Seq_Sort - Tool Sort, Closest Point **Select**

Tooling Requirements

☐ Make SubRoutines

☒ Develop Tooling Motions **Start and End Codes**
☒ Yes ☐ No

☐ Block Sequence

Information Development

☒ Make NC Code: ROUTER **Options...**

☐ Perform Machine Calc: Cut Time and Tool Changes **Select**

☐ Report Information: Router Report **Select**

☐ Edit Tooling Motions: Configuration Defined **Config...**

OK **Cancel** **Help**

NC Code Options

Postprocessor Name: ROUTER **Browse**

Output File Name: drawing1.out **Browse**

Job Identification Number:

Material X Dimension:

Material Y Dimension:

Material Thickness:

MHS Bunk Number:

☐ Manual origin ☒ Use line numbers

Starting line number:

Line number increment:

Generate MCU Comments
☐ Yes ☒ No

Preview NC Code
☒ Yes ☐ No

Program Comments: **Edit**

NC Code Generation Options
☐ Acceleration/Deceleration **Configure...**

OK **Cancel**

Feedrate

Feedrate

1000.00000

This field specifies the cutting maximum feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in.

Spindle Speed

Spindle Speed

18000.00000

This field sets the spindle speed in RPMs. This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Surface FPM and Units Per Rev.

Surface FPM

NONE

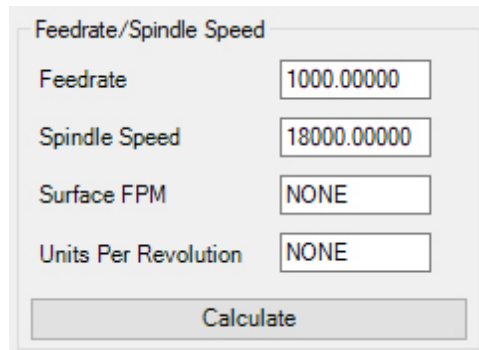
Units Per Revolution

NONE

These fields are useful for calculating feed rates and spindle speeds based on a tooling/material parameter referred to as Constant Surface Speed.

Surface Feed per Minute and Units per Rev are variables based on the size of each tool, that your tool bit manufacturer can supply you with.

The calculation of Feedrate and Spindle Speed can now take place using the Surface FPM and the Units per Rev fields. Simply input the Surface Feed per Min. and Units Per Revolution in their respective fields, click on the Calc button, and Router-CIM will input the correct feedrate and spindle speed automatically.



Feedrate/Spindle Speed	
Feedrate	1000.00000
Spindle Speed	18000.00000
Surface FPM	NONE
Units Per Revolution	NONE
<input type="button" value="Calculate"/>	

This Feedrate and Spindle Speed can be saved with your knowledge and used in future cutting operations.

NOTE: The actual settings input into the Surface Feed per Min. and the Units per Rev fields are not saved in the knowledge as they are used for calculation purposes only!)

Following is some information about Feeds and Speeds copied from the Onsrud Cutter© manual that may prove helpful in better understanding Surface Feed per Min. and Units per Rev.

To understand the concept of feeds and speeds, it is necessary to visualize what is occurring at the cutting edge of the tool. A chip of material is being removed from the base part. The size and thickness of the chip is controlled by the speed of the rotation speed of the spindle and the forward movement caused by feeding the tool into the material. If there is one flute, then the chipload is equal to the amount of travel in one revolution of the spindle. If there are two flutes, then there are two chips equal to one-half of the amount of travel in one revolution. If there are three flutes, then the chip load is one-third of the amount of travel in one revolution.

Most of the energy expended during these reactions is released as heat. Heat is one of the major factors in tool wear. The most effective way of getting rid of the heat is by having it carried away with the chip. This can be accomplished by cutting larger chips which both dissipate heat as well as yield a high quality part edge finish due to minimization of re-cut chips. This is possible if you have a tool that possesses a geometry that allows for both speed and finish characteristics.

After running a program, you can determine the actual feed value by timing one part or a total cycle time for a complete table of parts. The formula will be provided below.

There is another indication of proper feeds and speeds, and that is the tool temperature. After a run of parts, and after the spindle stops, check the temperature of the tool. If it is hot or warm to the touch, then the feed is too slow or the spindle speed is too high. If a proper speed and feed is used the tool should be at or near room temperature. Remember heat is what breaks down the cutting edge of a tool.

The first change to make is to the feed speed. This is the controlling factor in productivity. If the feed rate is at its maximum due to part configuration, hold down capabilities, software limits, or machine limitations, then the spindle speed should be lowered. This does two things; 1) It increases the chip

thickness and 2) It lowers the number of times the cutting edge is presented to the material. This second factor can be a major factor in increased tool life if this tool in this material has a limited number of cuts per sharpening. This could increase tool life by 15 to 20%. It also reduces the spindle bearing temperatures by reducing heat transmitted into the spindle.

NOTE: Please consult with your tooling supplier for their exact calculations. The below formulas are for example only.

Chip Load (Inches) = Feed Rate (IPM)/(RPM x No. of Flutes)

Feed Rate(IPM) = RPM x Number of Flutes x Chip Load (Inches)

Spindle Speed (RPM) = Feed Rate (IPM)/(Number of Flutes x Chip Load)

For Time Studies and True Average Chip Loads Use The Following:

Actual Feed Rate (IPM) = Circumference of the Part (Inches) x 60 Run Time (Seconds)

Ramp Feedrate

Ramp Feedrate

This sets ramp feedrates when a cut is doing multiple passes. The default is 1.0, Router-CIM's standard 100% feedrate for ramp down when doing multiple passes.

Note: Ramp Feedrate parameter will only affect CNC machines using G-Code as the NC code format.

Setting the parameter to a number less than 1.0 is a percentage of max feedrate set in the Control Panel.

Setting the parameter to a number greater than 1.0 will give you an exact feedrate.

Setting the Ramp Feedrate to 180 on a cut where the normal feedrate is 600, would allow the feedrate to be slower on the ramp down than normal. Normally the lead in would be 600 but in this case when a ramp is created between multiple passes in a cut, it will ramp down at a feedrate of 180 instead of the default 600.

You could also have set the Ramp Feedrate to 0.3 which is 30% of the programmed feedrate or 180 as well.

Before Codes and After Codes

Before Codes

After Codes

If a given code is needed at the beginning and end of a program (or part), such as a M08 (air blast on) and a M09 (air blast off), that code can be entered in the Before Code and After Code fields. This eliminates the need for inserting it manually in a text editor and the possibility of forgetting to add the code

Note: You must enter the codes correctly, for example M08 not M8. Please refer to the your Post Processor Application Notes for specific codes for your machine.

Oscillation Amt.

Oscillation Amount 0.00000

When selected, this option will create a tool path that varies its depth in the Z Axis constantly throughout the Cut. This ensures that the cutter will not be subjected to sustained frictional pressure at any one location, such as at the point where the cutter contacts a laminated surface.

Entering an amount in this parameter will produce a tool path that continually ramps either up or down until it reaches the depth given in the Total Depth field if the number is positive , or the depth of the Cut minus the Oscillation Amt if the number is negative.

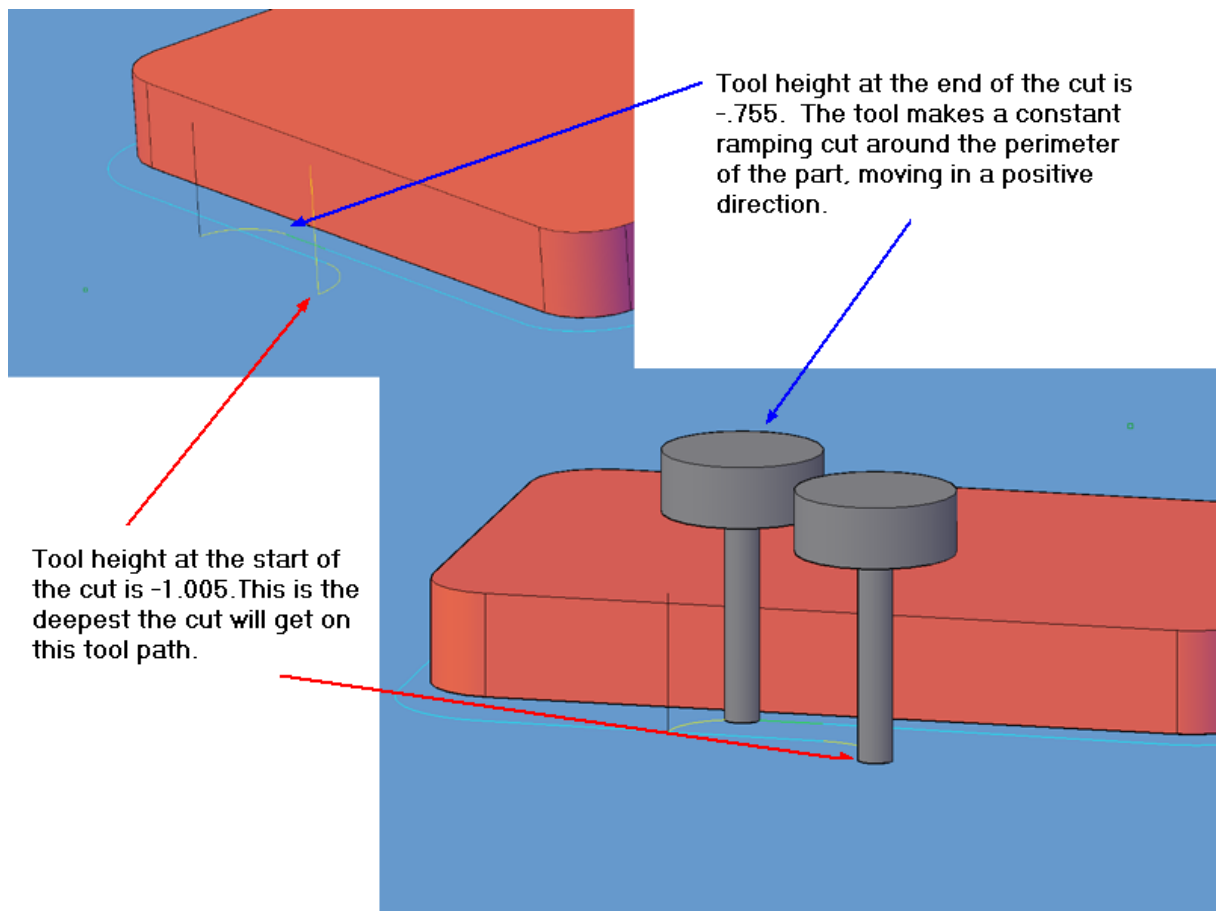
The following example will produce Cuts of various depths to allow for minimum tool wear and minimum removal of spoil board material.

NOTE: Oscillation amount ALWAYS has to have a number in this field. The field cannot be left blank.

EXAMPLE 1:

Material is $\frac{3}{4}$ " thick.

- Set Total Depth to -1.005
- Set Oscillation Amt to 0.25



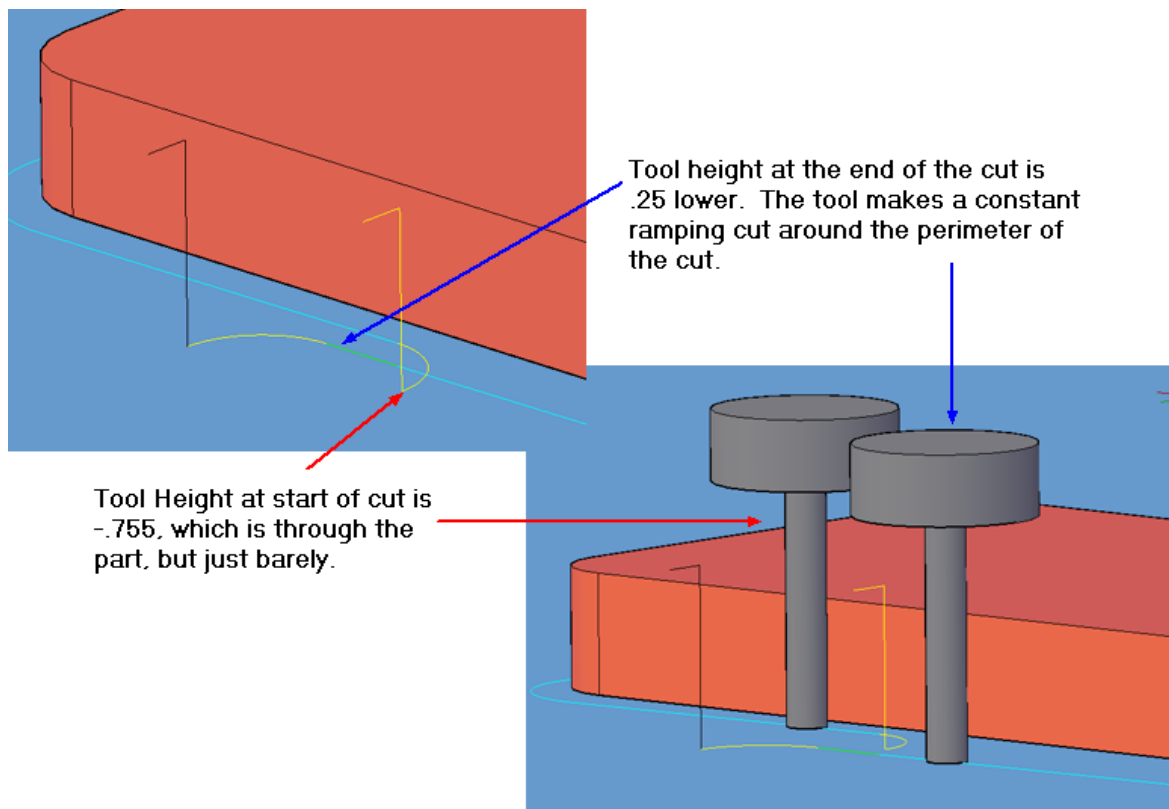
The cutter will enter the material and proceed to the Total Depth of -1.0 . It will then gradually lift up, in the Z Axis until it has reached a Z depth of $-.75$ by the end of the Cut.

Any laminate or other characteristic of the material that would have caused heavy wear to one portion of the cutter has been distributed over $\frac{1}{4}$ " of the cutting surface (which is the amount entered into the Oscillation Amt field).

EXAMPLE 2:

Material is $\frac{3}{4}$ " thick.

- Set TOTAL DEPTH to $-.755$
- Set OSCILLATION AMT to -0.25




The cutter will enter the material and start the Cut at -.755. It will then increase the depth until a total of -1.005 at the Cut end, and then retract.

Sort by Rank

Sort By Rank #

Rank is a numeric setting that allows tool paths and knowledges to be ordered in the code according to value, lowest to highest. When sorting by rank, the lowest rank number is cut first. You can make a tool path with rank 1, then another with rank 2. If you want a tool path to go between 1 & 2, make this tool path rank 1.5. Simply fill in a rank number before making a cut. Also note in the following sorting tasks, that area is defined as starting with the smallest and going to the largest.

Acc-N-Dec



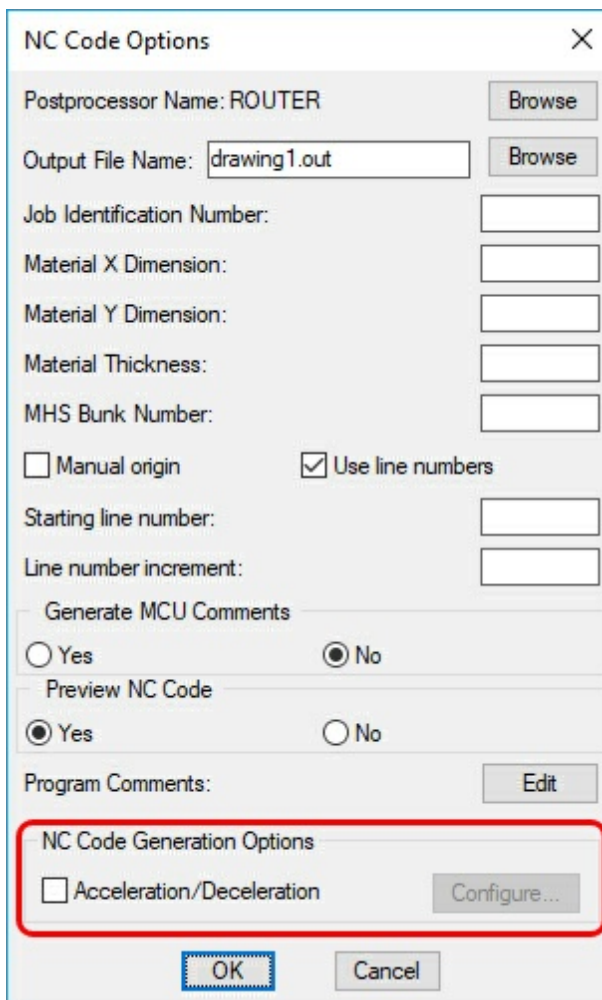
☒ Acc-n-Dec
☐ Metric
☐ Inline
☐ Plane Detect

DOIT File ...

Checking this box enables the Acc-n-Dec feature at the cut level. Acc-n-Dec will output feedrate changes in the code depending on the geometry the tool is cutting and the maximum feedrate programmed.

Check this box to apply Acc-n-Dec feedrate changes to the next tool path you make. To get the feedrate changes in the code, you would also check the Acc-n-Dec button in the Sequence Options.

For more information on configuring the ACC-N-DEC options, please refer to the [Acceleration/Deceleration](#) notes in the 'Make NC Code Options' section.



NC Code Options

Postprocessor Name: ROUTER

Output File Name:

Job Identification Number:

Material X Dimension:

Material Y Dimension:

Material Thickness:

MHS Bunk Number:

☐ Manual origin ☒ Use line numbers

Starting line number:

Line number increment:

Generate MCU Comments
☐ Yes ☒ No

Preview NC Code
☒ Yes ☐ No

Program Comments:

NC Code Generation Options
☐ Acceleration/Deceleration

If you wish to disable Acc-n-Dec later, de-select Acc-n-Dec in the Sequencer to skip insertion of feedrate changes for all tool paths in the program.

In the code example below, each of the feedrate changes have been highlighted.

```
%  
: 1 (ACCDEC SAMPLE 1)  
N1 G00 G17 G20 G28 G40 G80 G91 Z0 M5  
N2 G90  
N3 G52 X0 Y0 Z0  
N4 G08 P1  
N5 M08  
N6 (ROUTER-BIT .5 DIA.)  
N7 G28 G91 Z0 M05  
N8 G90 T2001 M06  
N9 T102  
N10 M03 S18000  
N11 G00 G17 G55 X43.9728 Y13.2444  
N12 G00 G43 H1 Z.25  
N13 G41 D01 G01 Y13.2559 F350.  
N14 Y13.5078 F500.  
N15 Y13.5194 F350.  
N16 Z.2385  
N17 Z-.4885 F500.  
N18 Z-.5 F350.  
N19 G03 X43.4728 Y14.0194 I-.5 F47.5  
N20 G01 X43.4014 F71.16  
N21 X25.5516 F1000.  
N22 X25.4801 F71.16  
N23 G02 X24.7311 Y14.7684 J.749  
N24 G01 Y37.9181 F1000.  
N25 Y37.9895 F71.16  
N26 G02 X25.4801 Y38.7385 I.749  
N27 G01 X62.2889 F1000.  
N28 X62.3604 F71.16  
N29 G02 X63.1094 Y37.9895 J-.749  
N30 G01 Y14.8398 F1000.  
N31 Y14.7684 F71.16  
N32 G02 X62.3604 Y14.0194 I-.749  
N33 G01 X43.4728 F1000.  
N34 X42.9728  
N35 G03 X42.4728 Y13.5194 J-.5 F47.5  
N36 G00 Z.25  
N37 G40 G00 Y13.2444  
N38 G28 G91 Z0 M5  
N39 G28 G91 X0 M09  
N40 G90  
N41 G52 X0 Y0 Z0  
N42 G08 P0  
N43 M30  
%
```

Metric



☒ Acc-n-Dec
☐ **Metric**
☐ Inline
☐ Plane Detect
DOIT File ...

Checking the Metric box will change the format of the NC code to metric mode instead of inch mode. **It will not change your feedrates or Cutter Compensation or your part size!** If you draw in metric, you should use METRIC tools, feedrates in millimeters per minute, etc. The purpose is to insert the code that changes the way the control reads the program from inch to metric.

On most controls, the metric G-code is G21. The inch G-code is G20. Some controls use different G-codes and your post processor should output the code appropriately.

Inline



☒ Acc-n-Dec
☐ Metric
☐ **Inline**
☐ Plane Detect
DOIT File ...

This option allows you to add specific commands to your code during the Cut phase. If this box is checked, when you make a Cut, you will be prompted with a question...**Put Commands In Shape?**

You can then select either Yes or No. If you select Yes, you will then be prompted to Choose a Point Location: select a point on the geometry where you want to run, edit or insert a given task or command.

Next you will be prompted to Select/etc.... or <Exit> (Type in Select if selecting more than one point or press <Enter> to exit.

You will then be prompted to either RUN A TASK, EDIT A TASK, or INSERT A POST COMMAND.

If you select RUN A TASK, you will be prompted to enter the TASK NAME. This is usually only done if you have a specific TASK FILE (usually provided by CIM-Tech) to run.

If you select EDIT A TASK, you will be shown the TASK EDITOR, and you will be able to select the TASK you wish to modify. Again, this is usually on a specific TASK, and is not generally needed.

The final option is to INSERT A POST COMMAND. When you choose this option, you get a SELECT POSTPROCESSOR COMMAND dialog box with several POST COMMANDS that you may INSERT into the drawing, at locations you choose. Click on the COMMAND you wish to insert then <OK> and you will either be prompted for a value or you will be prompted again as to whether or not you want to Put Commands In Shape?...

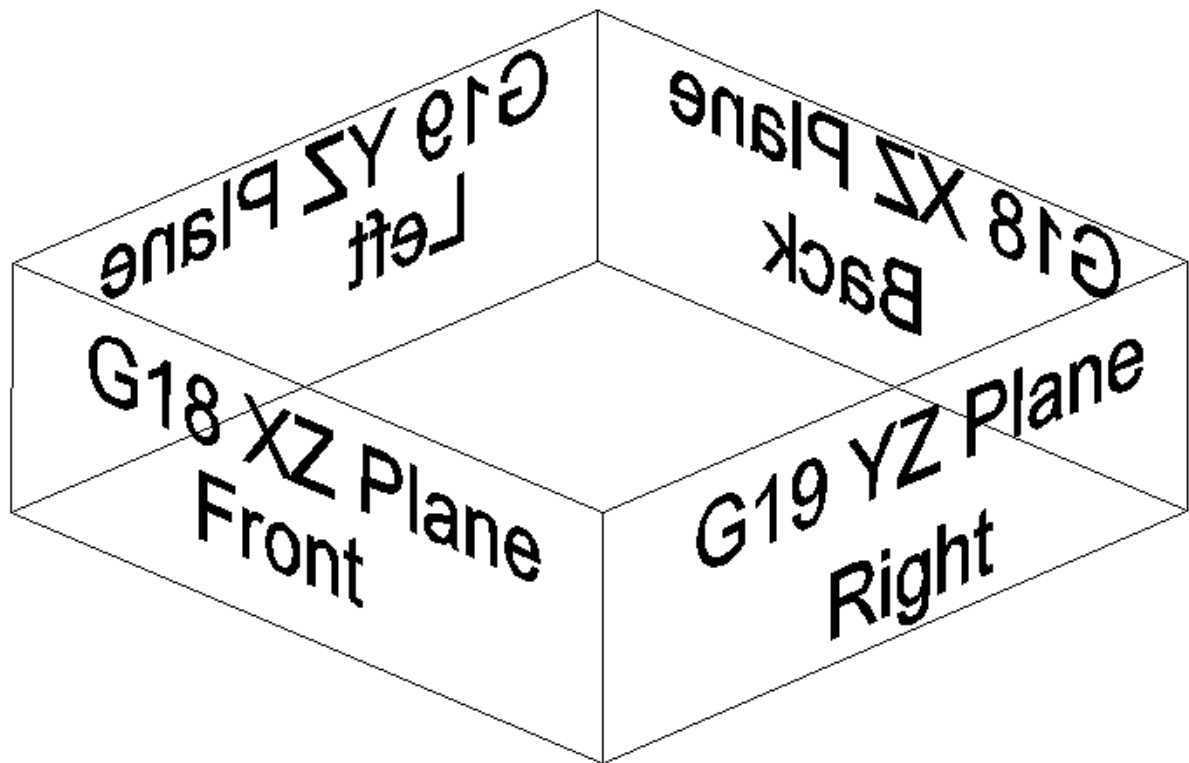
This is useful if you need to insert a program stop, or a programmed pause into the Cut.

NOTE: INLINE does not work with TABBING, or OSCILLATION.

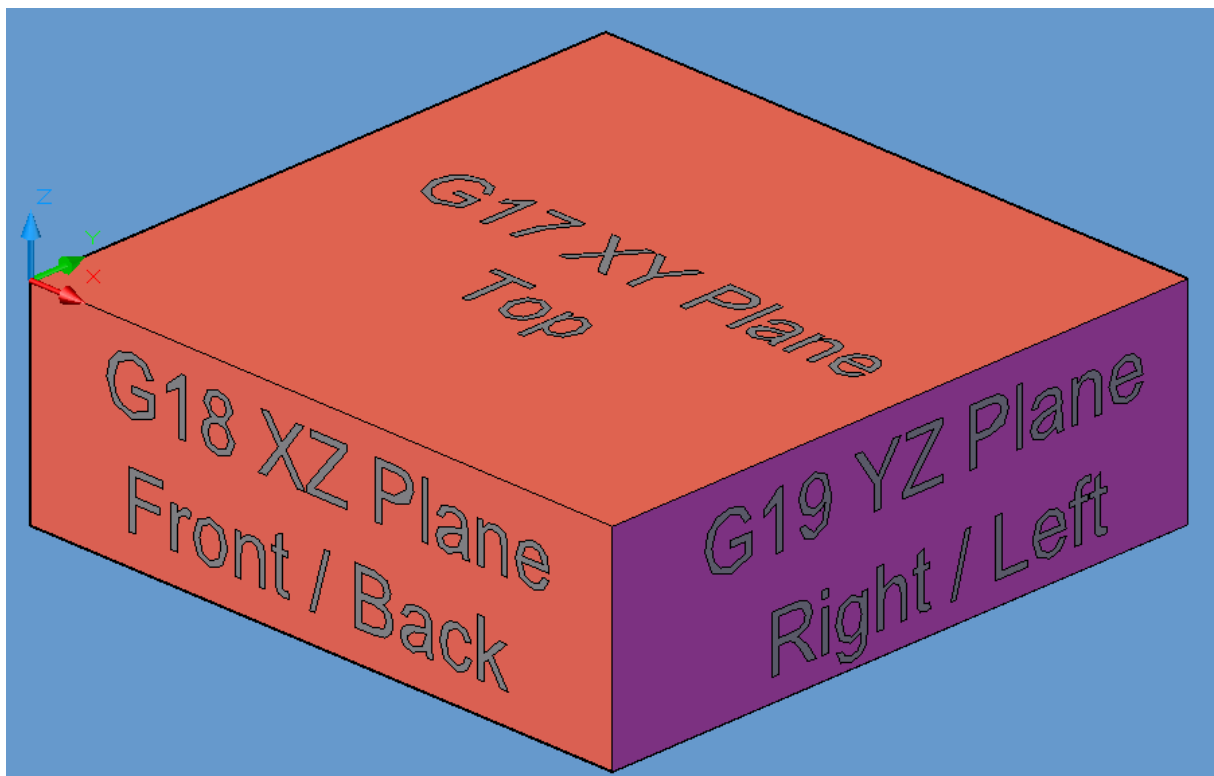
Plane Detect



This parameter is used to check whether or not the cut is in the XZ (G18) or YZ (G19) plane. By checking the box, Router-CIM will produce the most efficient code possible for geometry outside of the XY plane. This is accomplished by automatically evaluating the tool path face (Coordinate System) for its eligibility to use Arc Interpolation vs. Point to Point. Plane detect will be reflected in your code by either a G18 (XZ plane selection) or G19 (YZ plane selection) if possible, or G17 (XY plane selection) otherwise.

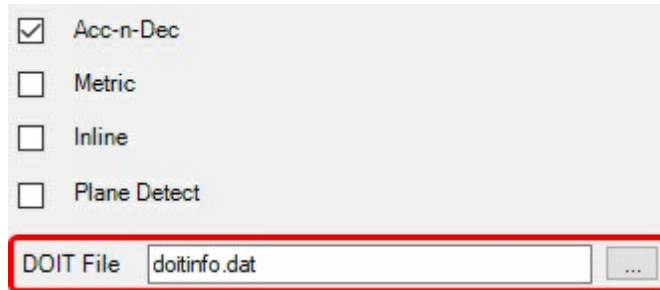


This drawing illustrates the fact that the front and back are both on the XZ plane and the right and left sides are both on the YZ plane.



The G17 plane is the XY plane. If any horizontal cuts do not lie exactly on the G18 or G19 faces, then they will be output as G17 code.

DOIT File



The screenshot shows a control panel with four checkboxes: ☒ Acc-n-Dec, ☐ Metric, ☐ Inline, and ☐ Plane Detect. Below these is a text field labeled "DOIT File" containing the text "doitinfo.dat". To the right of the text field is a small button with three dots "...". The entire text field and button area is highlighted with a red rectangular border.

This field allows you to call up or choose which .dat file you want to make current for the DO-IT cycle. When you have several DOIT files and wish to edit one, type the name into the DOIT File box and then click on the DOIT Edit button at the top right of the Control Panel/All Stats page.

The button to the right of the entry box is to allow you to browse your computer to select an existing file.

Cutting

After selecting the Tool, Cycle and Status Information necessary to make a Cut, you must then select Cut from the Control Panel (or type Cut at the command prompt or select it from the Router-CIM toolbar) and you will be prompted to select objects. Pick the shape you wish to Cut and using the current settings from the Control Panel, a tool path will be generated using the parameters you have defined.

A Tool path will show up either as green (clockwise) or red (counter-clockwise) along with the Lead-In and the Lead-Out as yellow and the Rapid Moves in white.

The tool path will be generated at the Z depth you specified in the Total Cut Depth and/or Depth per Pass fields.

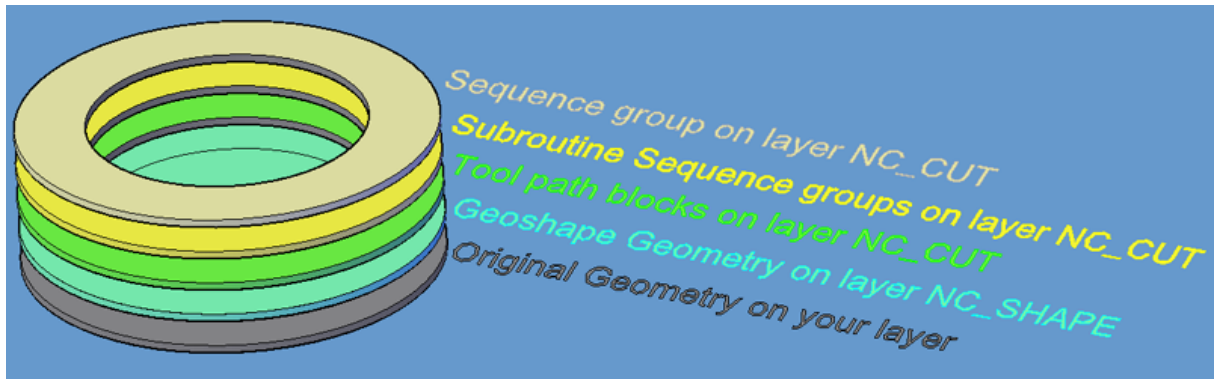
You may only Cut Polylines and surfaces that are on the **NC_SHAPE** layer.

Layer Control

Be aware of the Layering scheme that Router-CIM creates when tool paths are created.

The items on the screen will be on top of each other in this order:

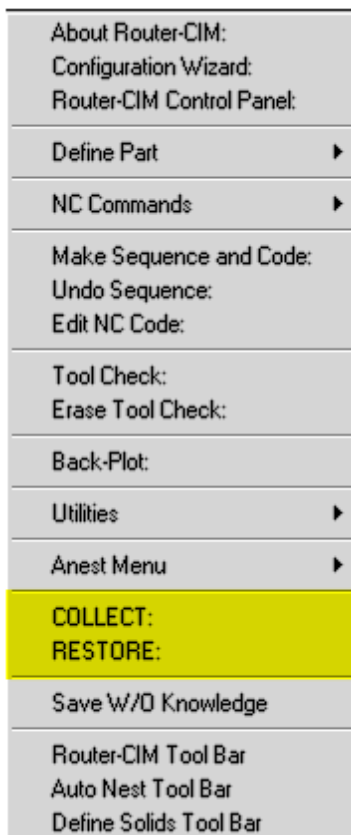
1. Your original geometry on whatever layer you built it.
2. Defined CCW Polylines from the geoshape command are on layer NC_SHAPE.
3. Tool Path blocks created from the CUT command are on layers starting with NC_Cut.
4. The Sequence List for each tool are on layers starting with NC_Cut, if using Sub Programs.
5. The Sequence List for all tool paths on layers starting with NC_Cut.



Collect and Restore

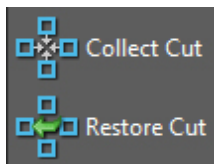
When CUTS have been made and you want to edit or redo some of the CUTS, it is sometimes difficult to see one Cut underneath another for the purpose of selection.

Select the RCIM pull down menu, COLLECT and RESTORE will appear.



OR

Router-CIM Ribbon:



Keyboard: **COLLECT**
 RESTORE

To use Collect:

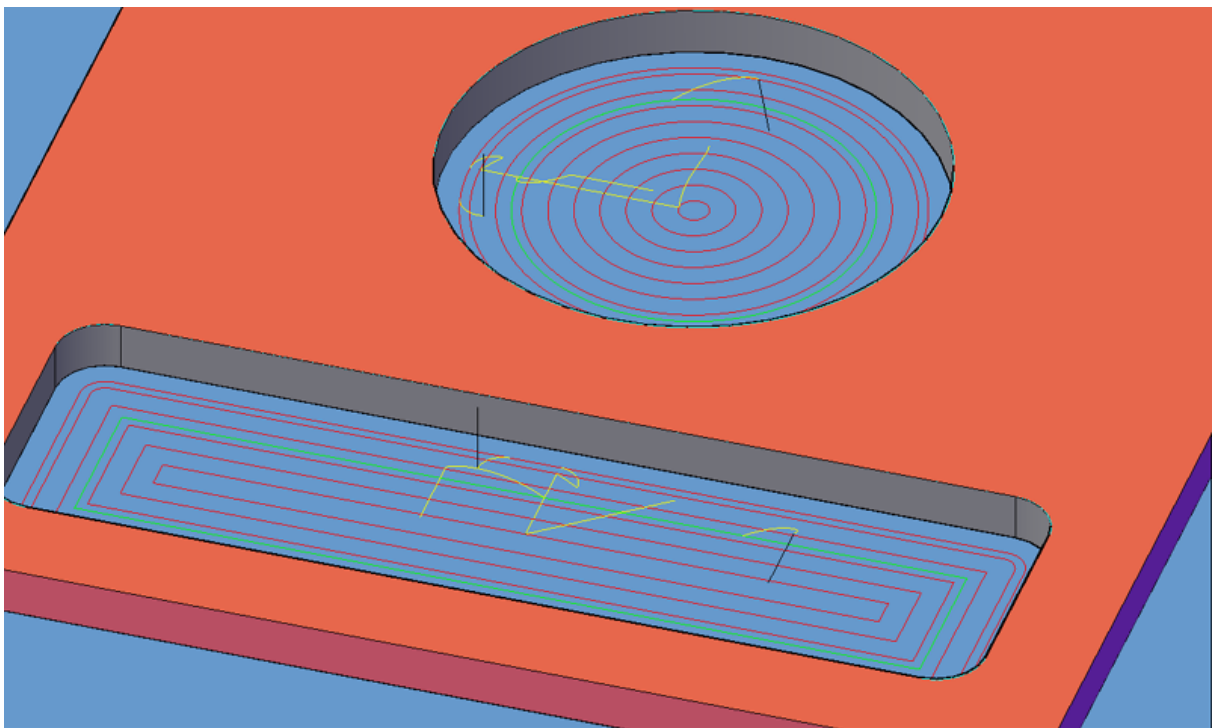
Simply select COLLECT from the menu, and pick on the tool path you want to remove from the screen temporarily. It is collected to a temporary buffer in memory, available only as long as the drawing is open. The RESTORE command will clear the buffer and display the Cut on the screen once again.

To use Restore:

Select the Restore command from the menu, and any cuts that have been Collected with the Collect command will be Restored to the drawing.

Using Collect and Restore with Sequence to create a Sort Order.

If the Cuts are Collected according to the order in which you wish them to be cut, then they can be Restored in order. If you immediately use the Sequence command, you can use the P for Previous option when prompted to Select Objects for the Sequence. The order the cuts were Collected in will be the order that they are stacked in the Sequencer, and no further sorting will be necessary.

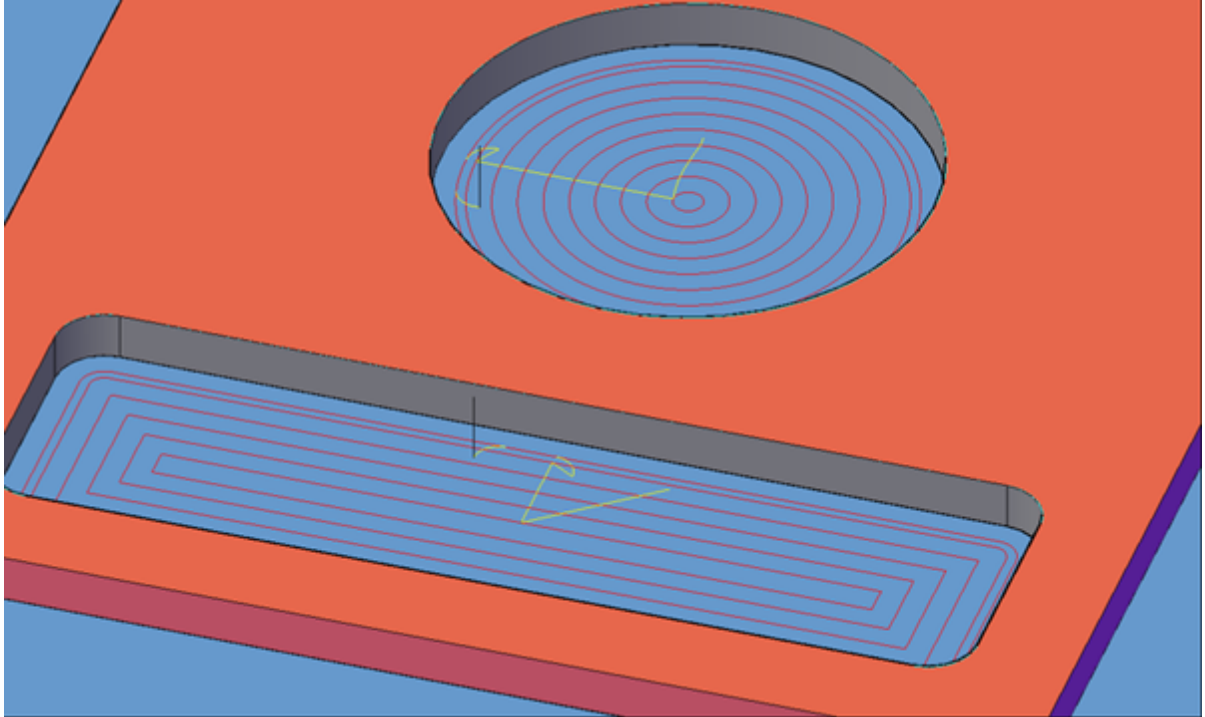


Part with multiple tool paths

For example, if in the following cuts you wish to use MULTI-PASS to cut the inside of the part first and then cut the inside shape all the way through with another cycle, you should pick on the MULTI-PASS cut first to COLLECT it, and then COLLECT the other inside cut. If no other action is taken between

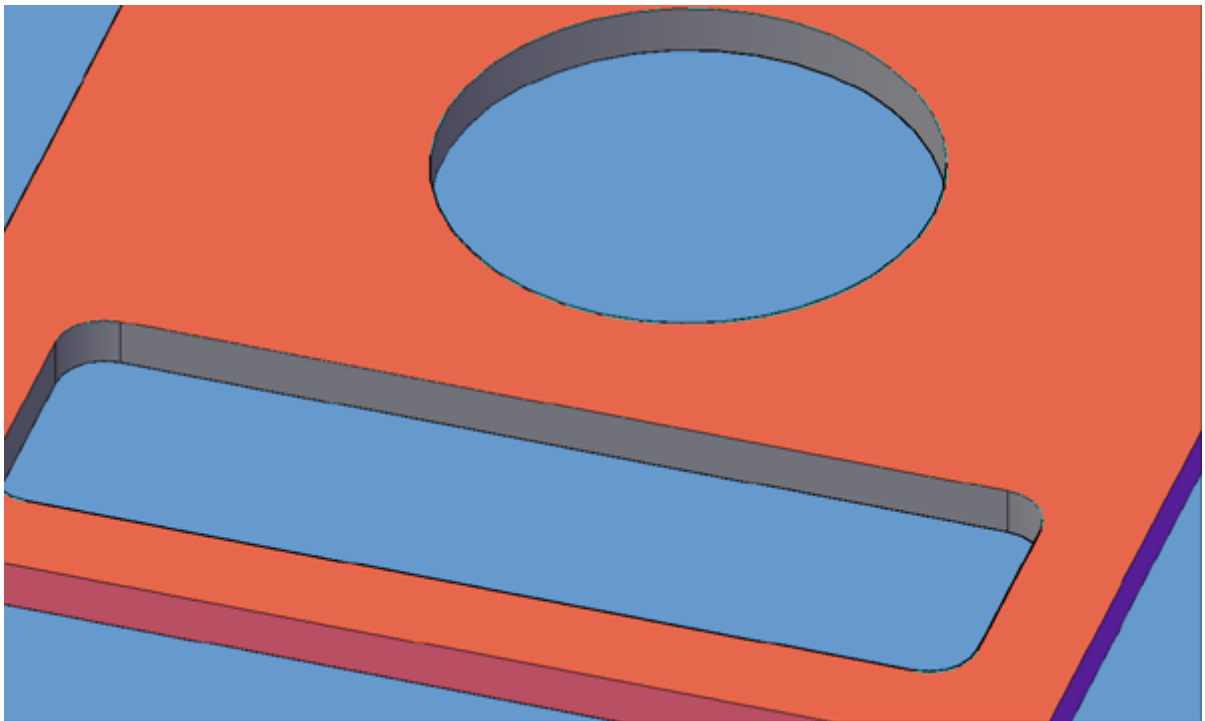
COLLECT and RESTORE the cuts will be restored in the order in which they were collected. If Sequence is the very next step, the Previous selection set will select the cuts in the same order, eliminating the need for further sorting.

Make the First Cuts



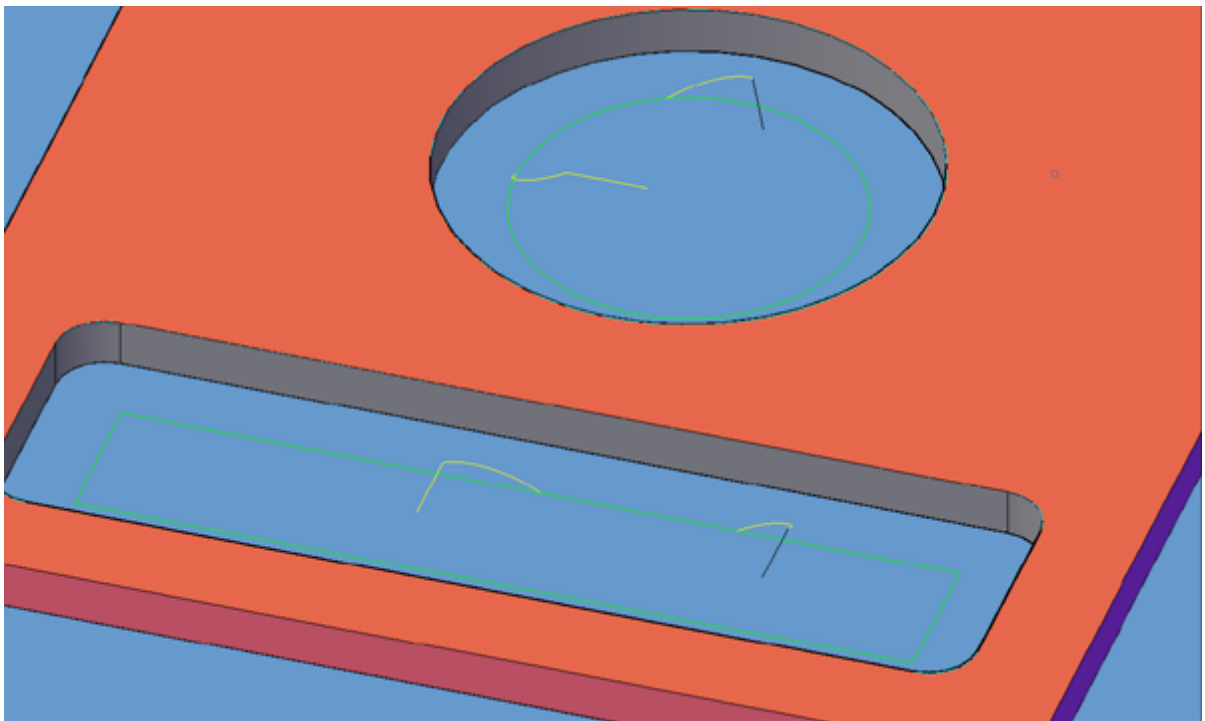
Multi-Pass tool paths created

Then COLLECT them in the order you want them cut. The tool paths will be removed from the screen.

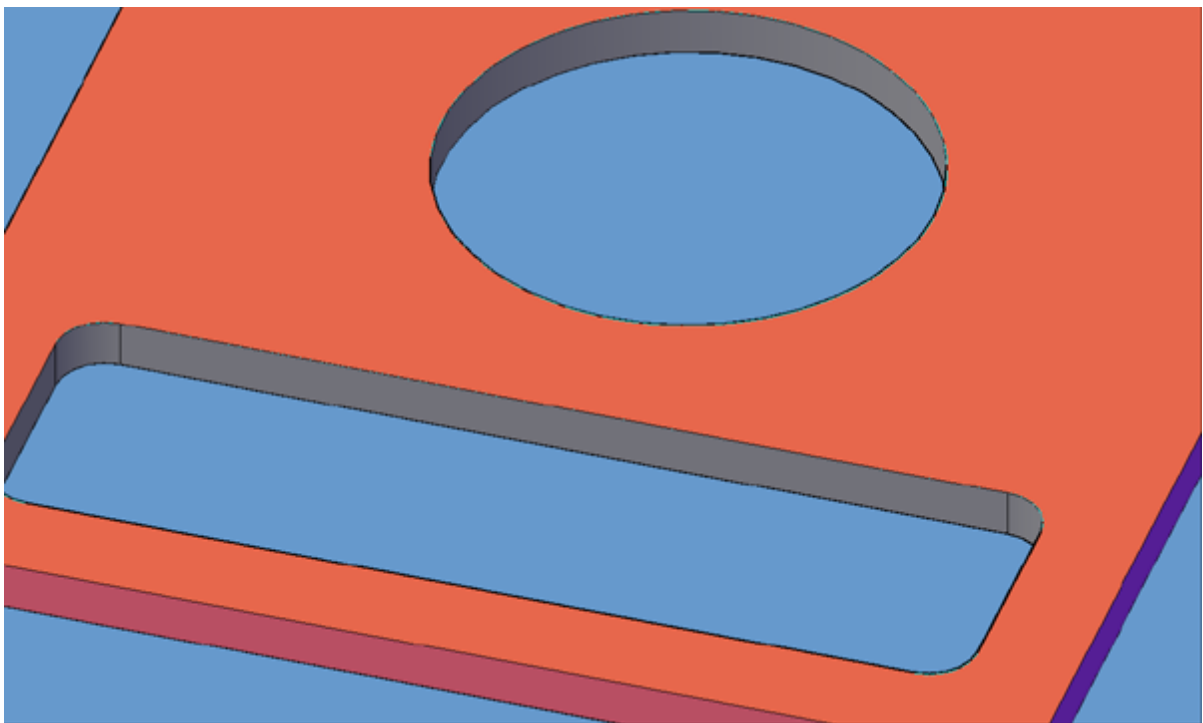


tool paths Collected

Next make the Shaper Tool cuts on the part.

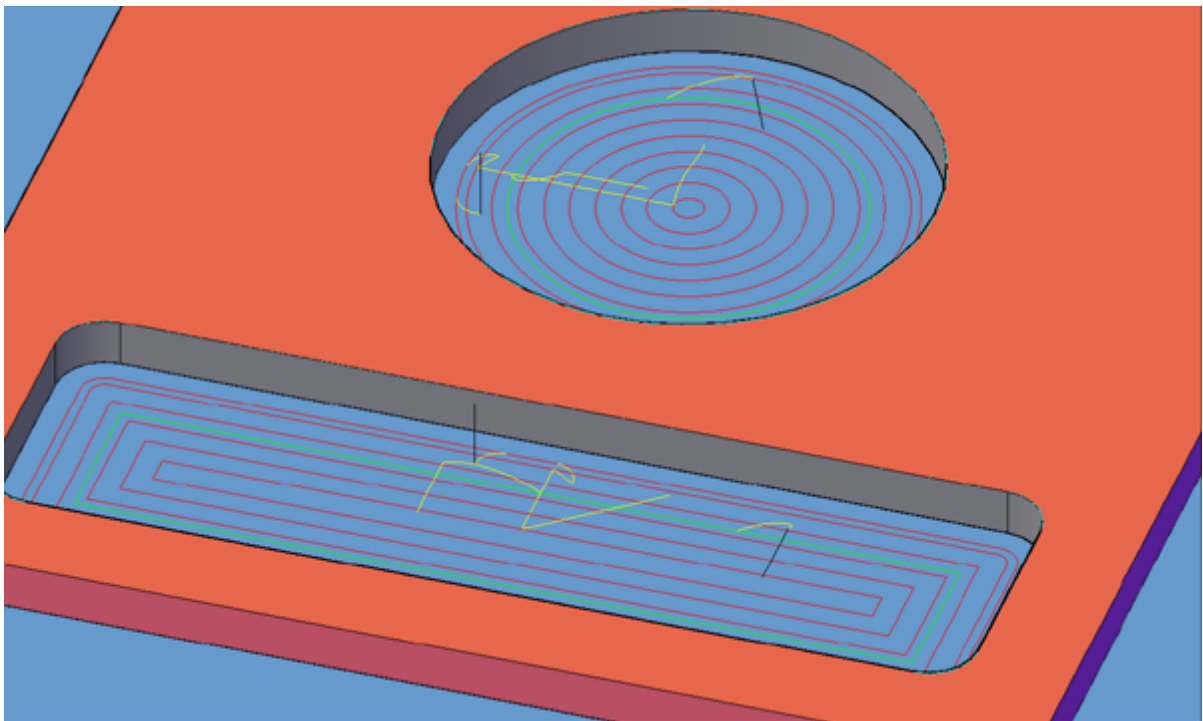


Then COLLECT these cuts in order.



Tool paths Collected again.

Now, RESTORE the tool paths.



Restore the tool paths.

Finally, select Sequence and when prompted to Select Objects, type in P (for Previous) and then make your code. No other sorting tasks are necessary and the cuts will be Sequenced in exactly the order they were Collected in.

Cutting Text (LetterEase)

Cutting text involves the use of a tool called LetterEase. This tool can be selected from the Router-CIM toolbar. Letterease allows you to select different text styles, and then explode them into lines and arcs so that you can Geoshape and cut.

Router-CIM Toolbar:

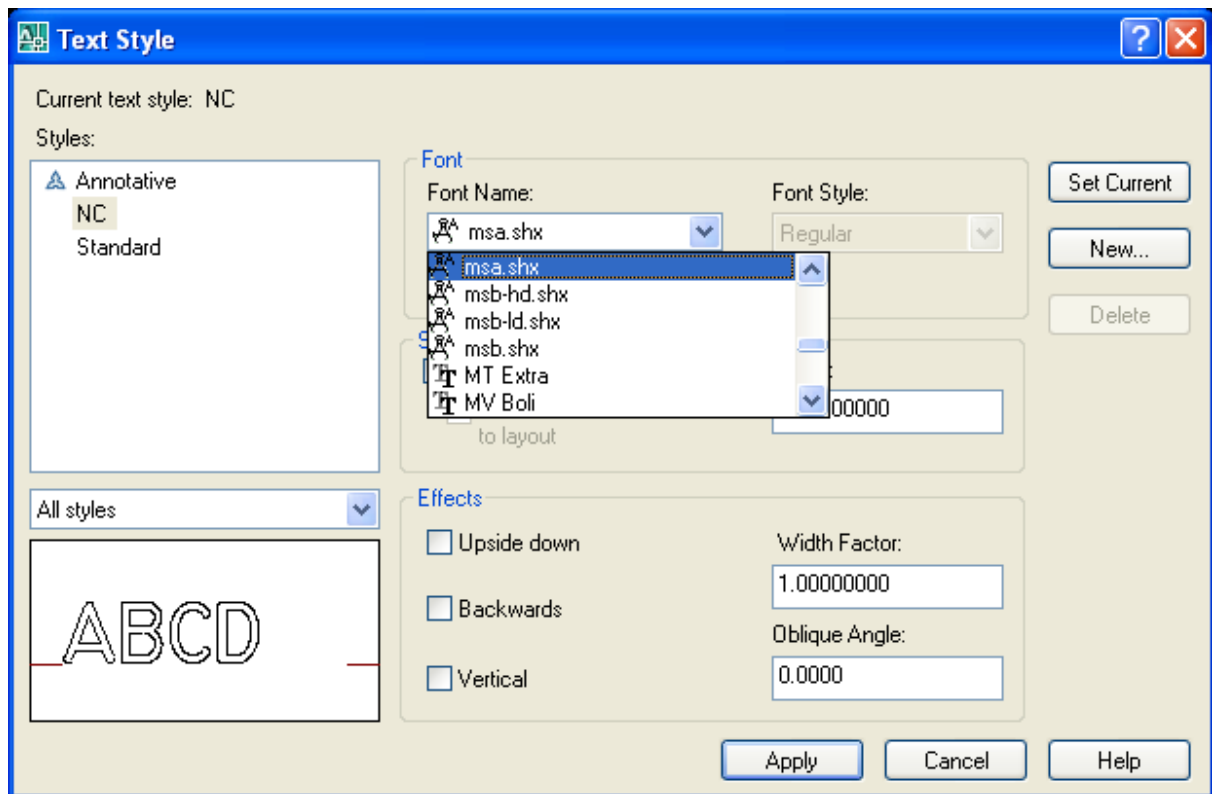


Router-CIM Ribbon:



Keyboard: **LE**

To use the standard Fonts that come with AutoCAD, select a Text Style.



From the Text Style box, you can preview and select a new text style, set the height, width, angle, etc. for your new lettering.

Next apply the lettering to the drawing with the DTEXT (Draw > Single Line Text).

```
Specify start point of text or [Justify/Style]:  
Specify rotation angle of text <0.0000>:
```

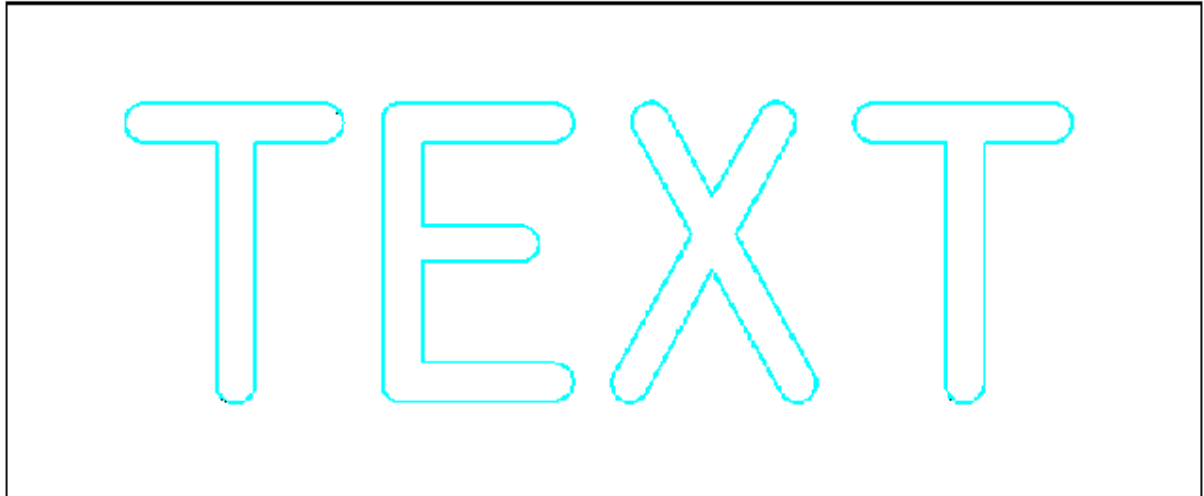
You can Geoshape this text, after you explode it. To explode the text, pick the LE button on the Router-CIM toolbar.

```
Command:  
Baseline object/Circular/Edit/Justify/Spacing/eXplode/<Start point>:
```

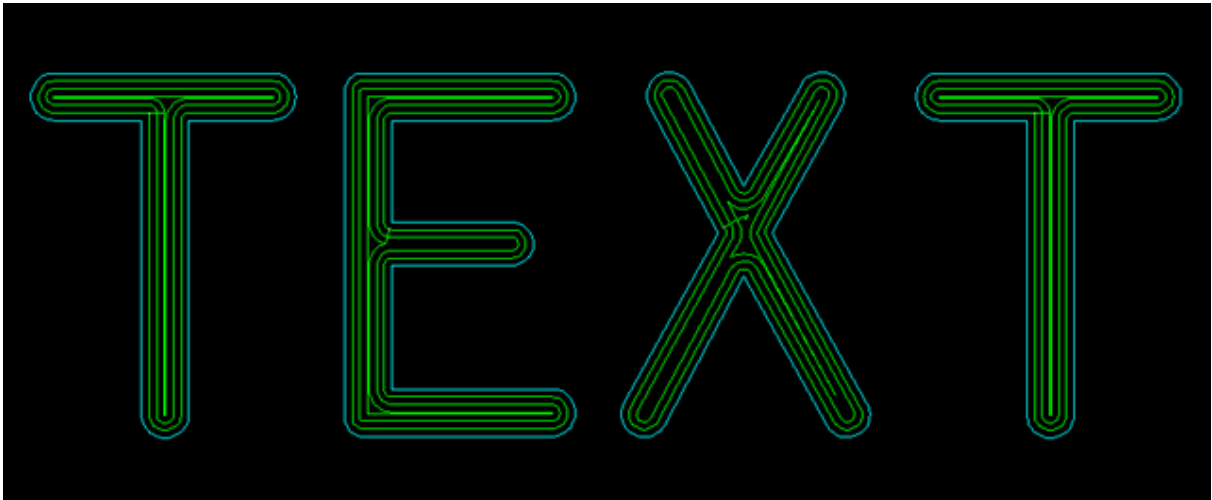
Type X for Explode to convert the text to polylines and polyline arcs.

```
Exploding Text Entry object 1....Done!  
Command:
```

Once the text is exploded, you can Geoshape the text to make it ready for cutting.



Once the text is Geoshaped, select a tool, cycle and set the status information to create tool paths.



Baseline Object

This option will allow you to place text along a polyline, arc or circle on the screen. These methods are explained in detail below.

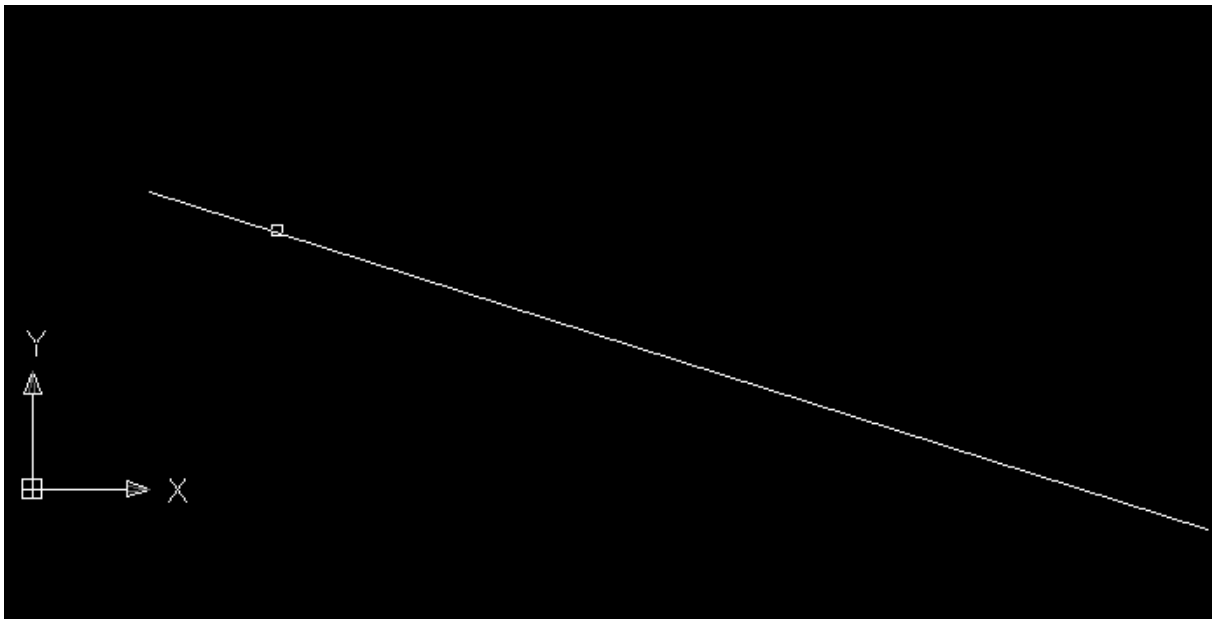
Baseline Object - Polyline

To use a Polyline object, you can examine the following explanation:

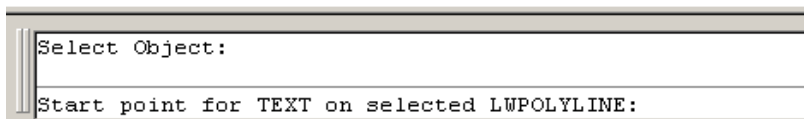
First you should set the text style to the style you want the text to use. Then type 'B' for Baseline object and you will see the following prompt:

```
Select Arc, Circle, Polyline:
Select Object:
```

You should select the polyline you want the text aligned to:

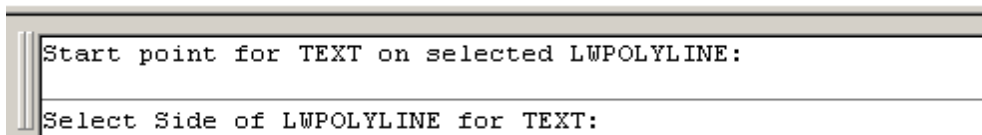


Then you will be prompted to pick an Start Point on that polyline where you want the text to start.

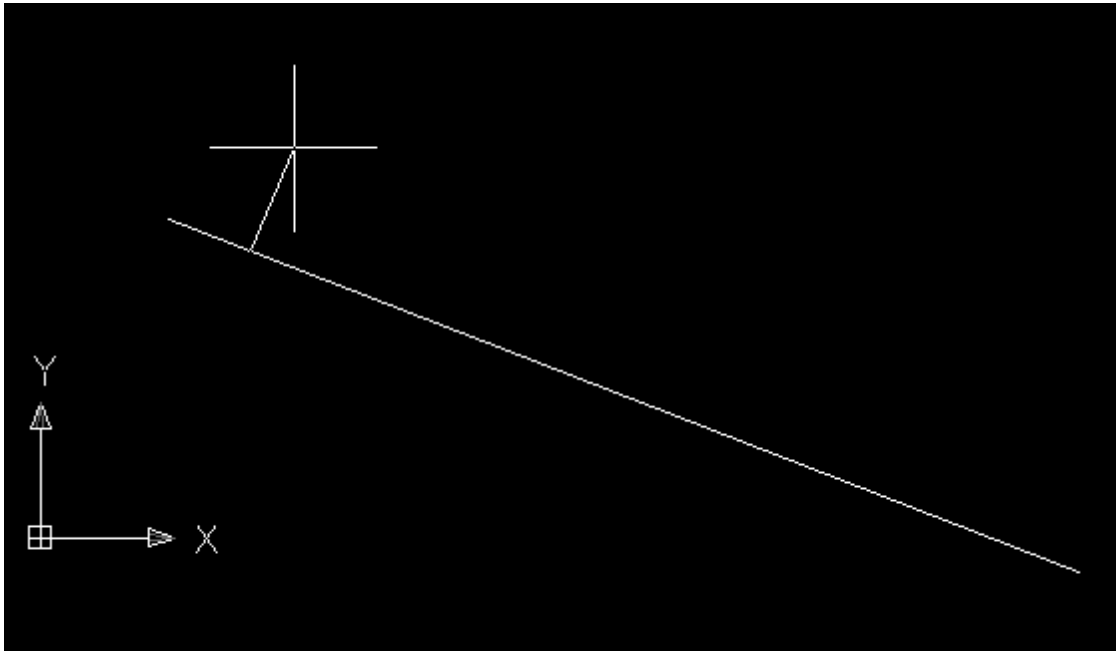


You will see the nearest snap appear on your polyline and you can then pick a point where you want the text to start.

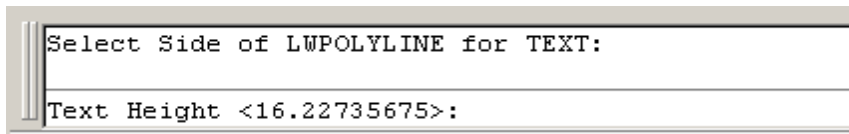
Next you will see a prompt to pick the Side of the polyline you want the text to appear on.



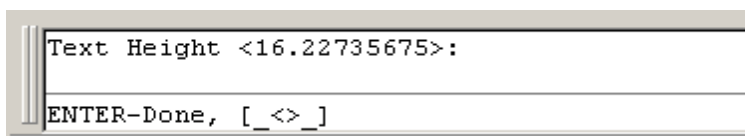
You will have a rubberbanding line extending to your cursor and you can then pick either side of the polyline for the text to appear on.



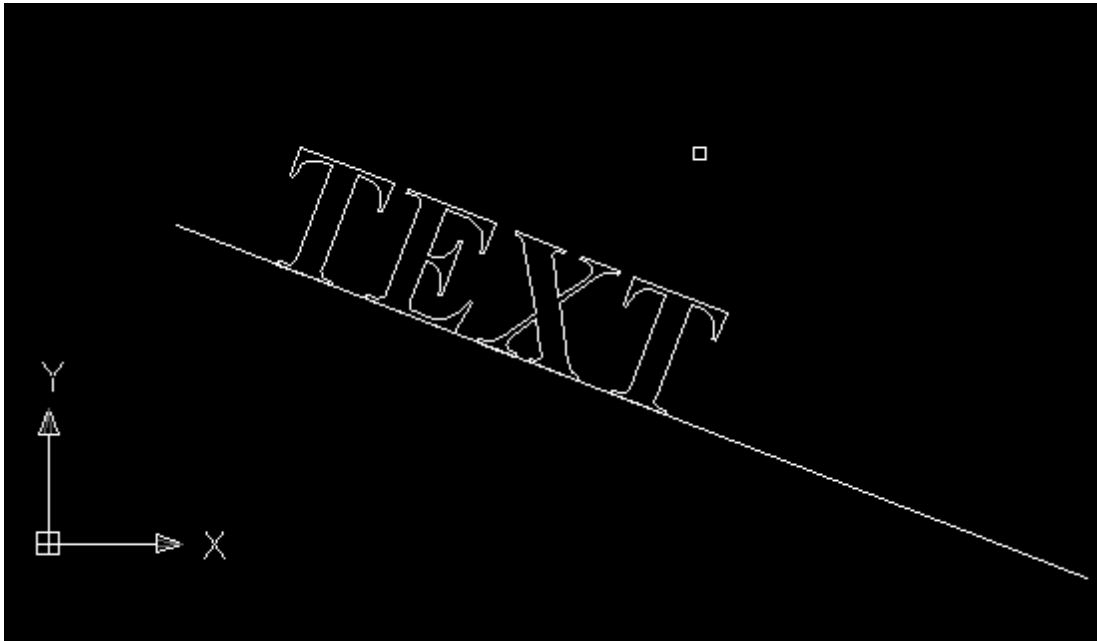
After picking the side you want the text to appear on, you will be prompted to select the Height of the text.



You can either pick a point on the screen for the text height, or type a number onto the command line for the text height. Once you select a value, you will then be able to enter the text in that you want to appear on your polyline when you see the following prompt:



Type in the text you want and press Enter when you are finished and the text should appear on your screen aligned with your polyline.

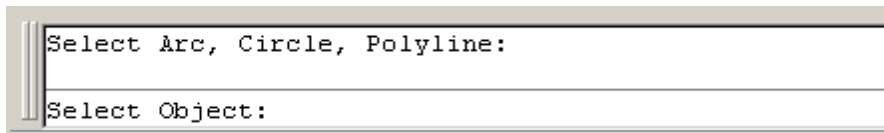


You can then use the LE button and explode the text and cut it as normal.

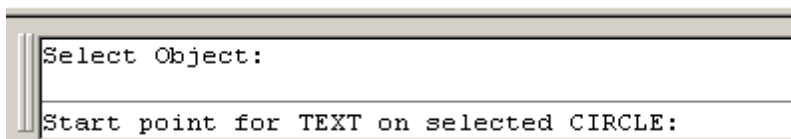
Baseline Object - Circle

You can use a circle to serve as a base for aligning your text.

When you select the Baseline object and are prompted to select arc, circle, or polyline, select the circle that you want the text aligned to.

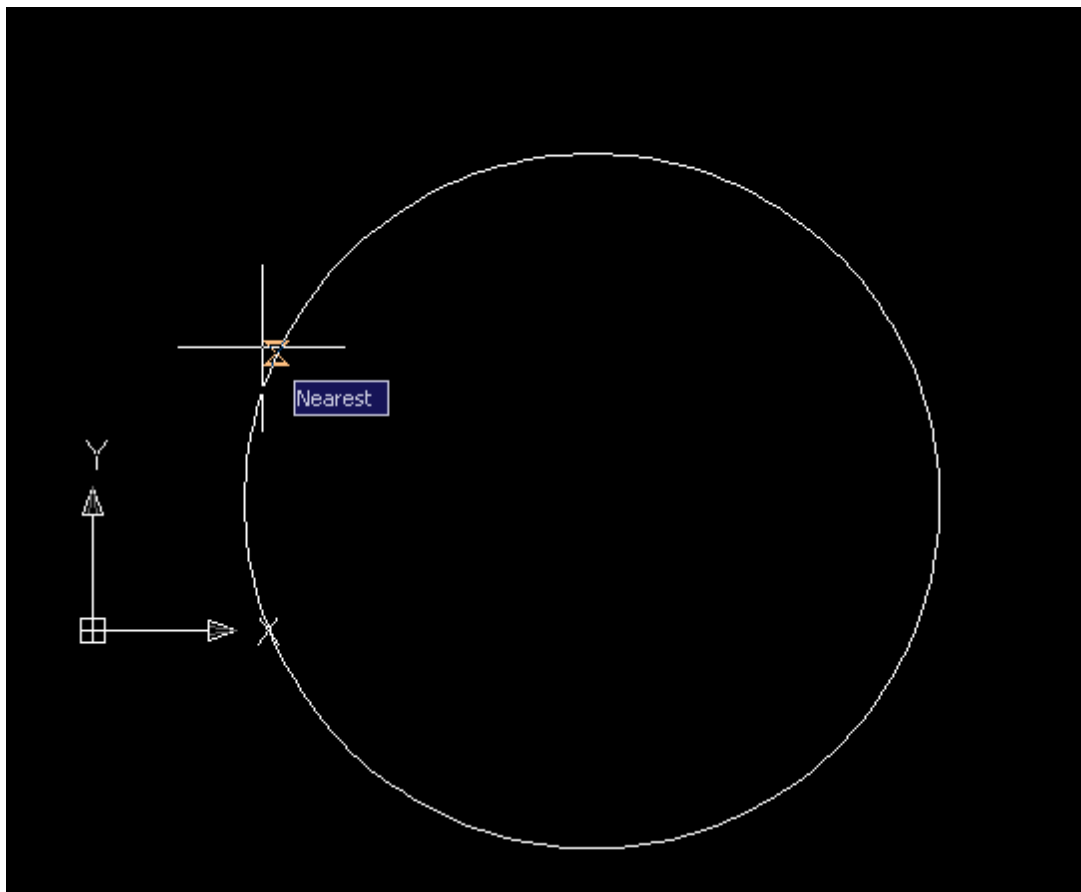


Next you will be prompted to select a Start Point on the circle for your text to start at.

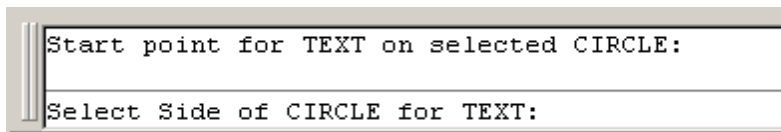


This can be a rough point along the circle as you will get a chance to center the text about the circle later.

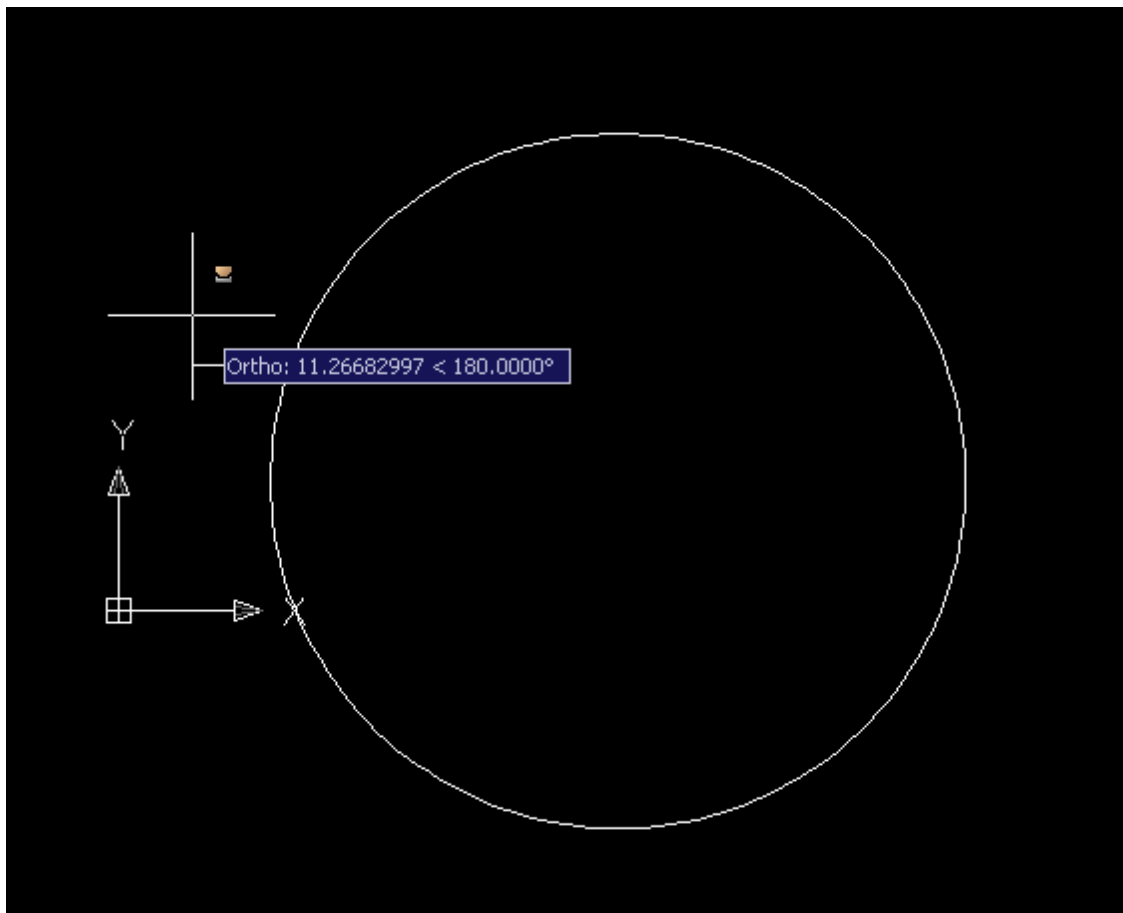
Select a location along the circle.



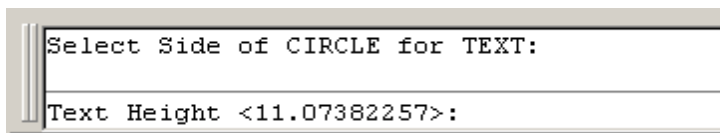
Next you will get a prompt to pick the Side of the circle you want the text on.



Pick either the inside or outside of the circle.

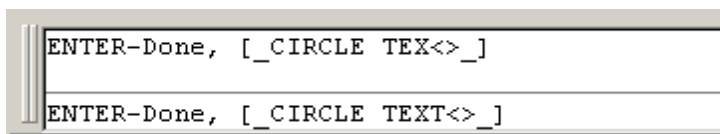


Next you will be prompted for the Height of the text.



You can either pick a text height on the screen or type in a value for the height at the command line.

Once the height is established you will see the following prompt.

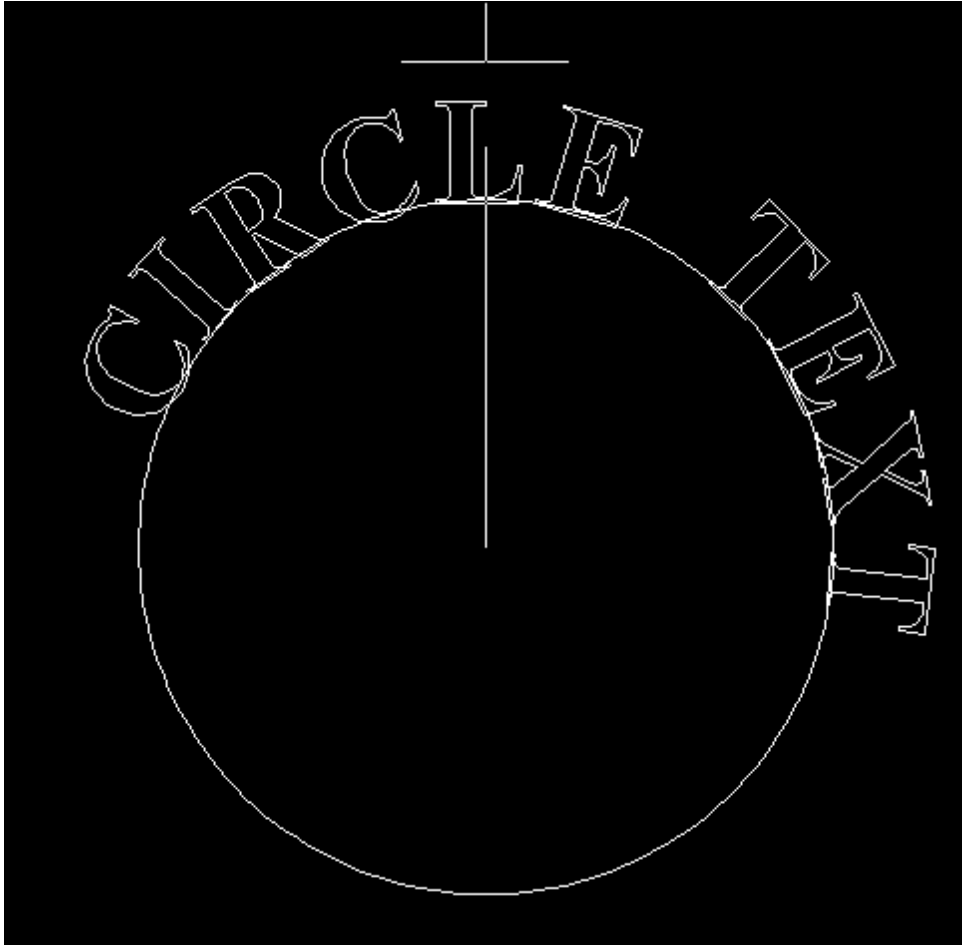


Enter in the text you want to appear on the circle and then press ENTER when finished. You will then get a prompt to pick the Angle to center the text around.

```
ENTER-Done, [_CIRCLE TEXT<>_]
```

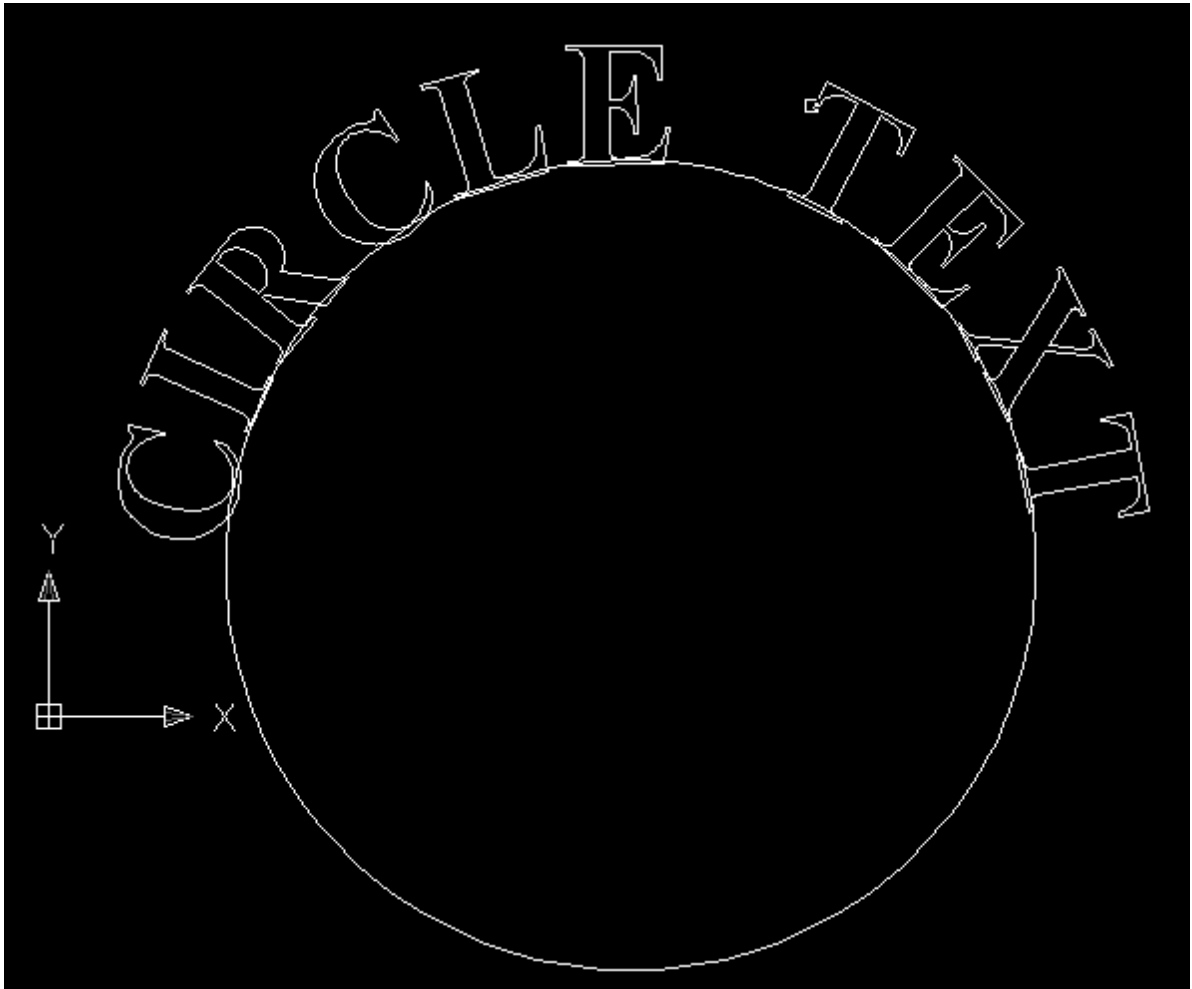
```
Angle to center text around:
```

and you will have a rubberband line from the center of your circle to your cursor.



Selecting this point will allow you to have all the text spaced out to be centered around the point you pick.

In this instance the 12:00 position was selected.



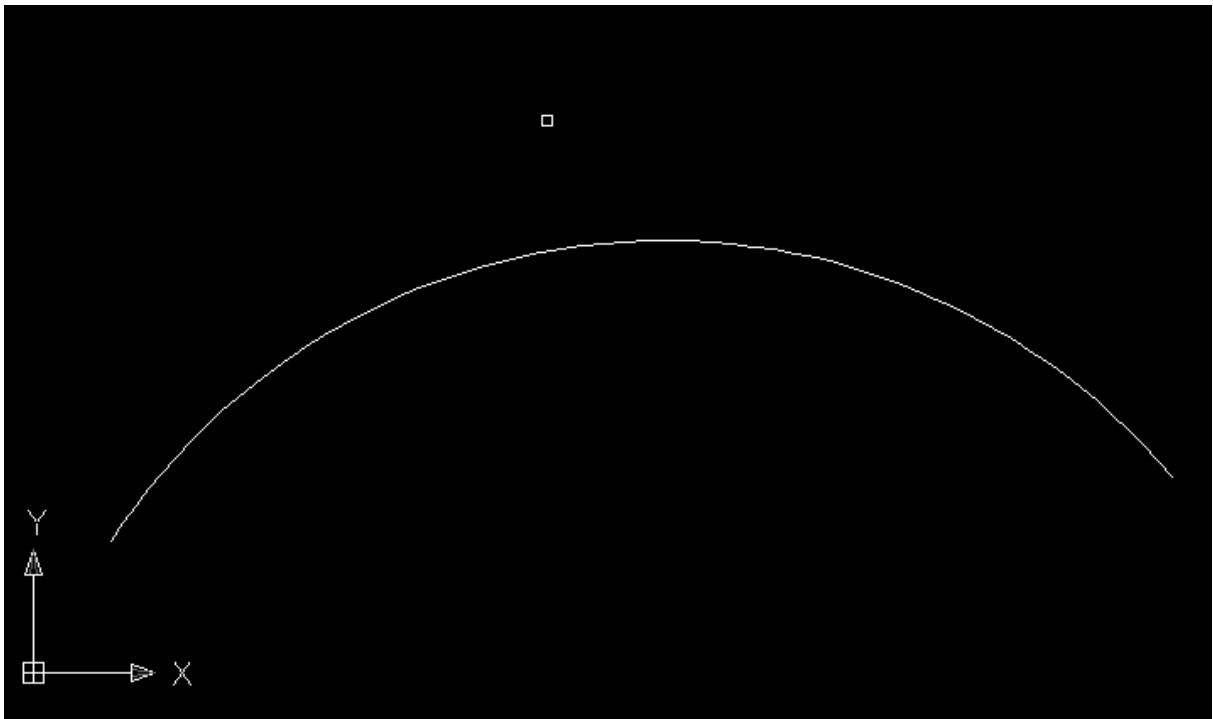
And the text was shifted so that it was centered around that point.

You can now explode the text and cut it as necessary.

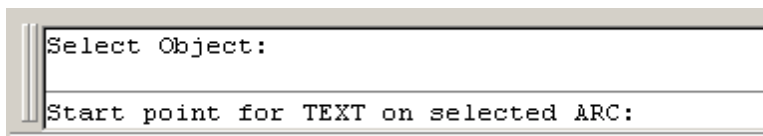
Baseline Object - Arc

The Arc aligned text is the same as the Circle aligned text and contains the same features.

First, when selecting Baseline object, you pick your arc.

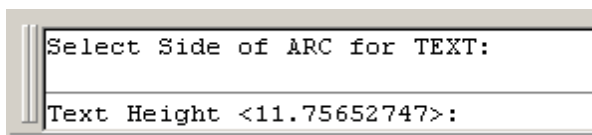


You will then get a prompt to select the Start Point along the arc.

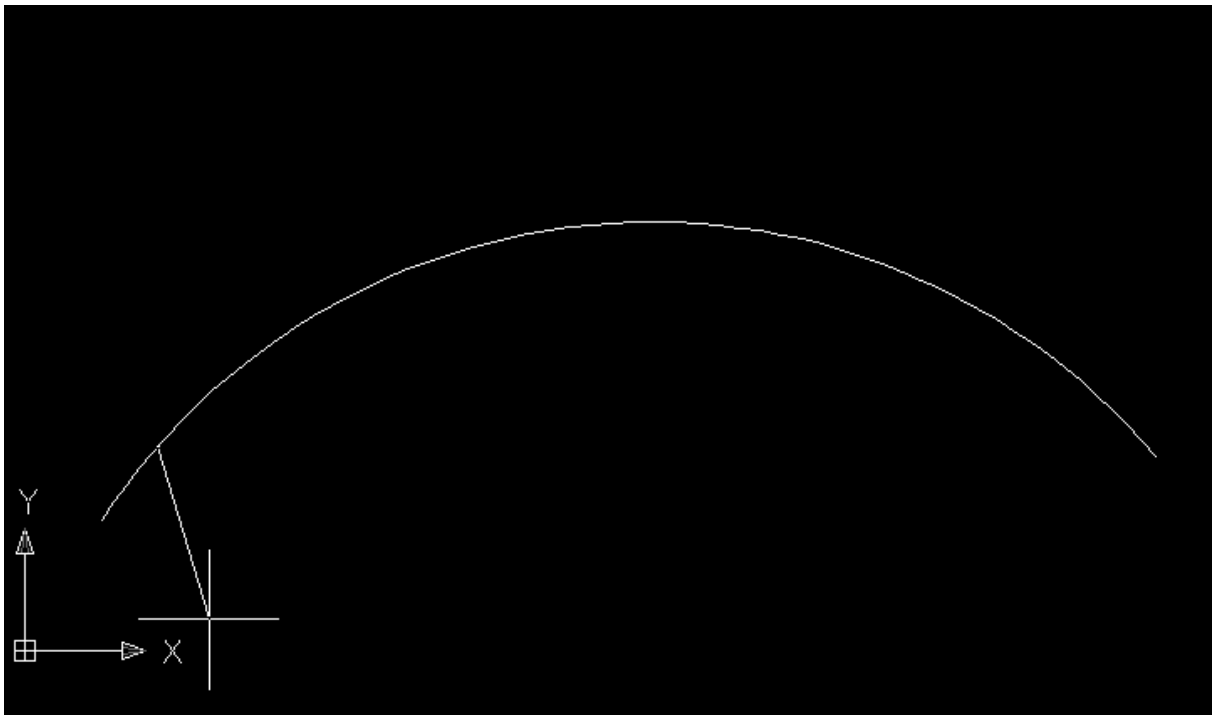


Pick a point on the arc where you want the text to start.

Next you will be prompted for the Side of the arc you want the text on.



Pick either the outside or inside of the arc. You will have a rubberbanding line attached to the cursor to show the side selected.



Next you will get a prompt for the text height.

```
Text Height <11.75652747>:  
ENTER-Done, [_<>_]
```

You can either pick a text height on the screen or type in a value for the height at the command line.

Once the height is established you will see the following prompt to enter in the text you want on the arc.

```
ENTER-Done, [_ARC ALIGNED<>_]  
Angle to center text around:
```

Type in the text you desire and then press ENTER and the text will appear on the arc.



You will be prompted to pick the Angle to align the text by.

Select the point where you want the text centered around. In this case a point at the top quadrant (12:00) was selected.

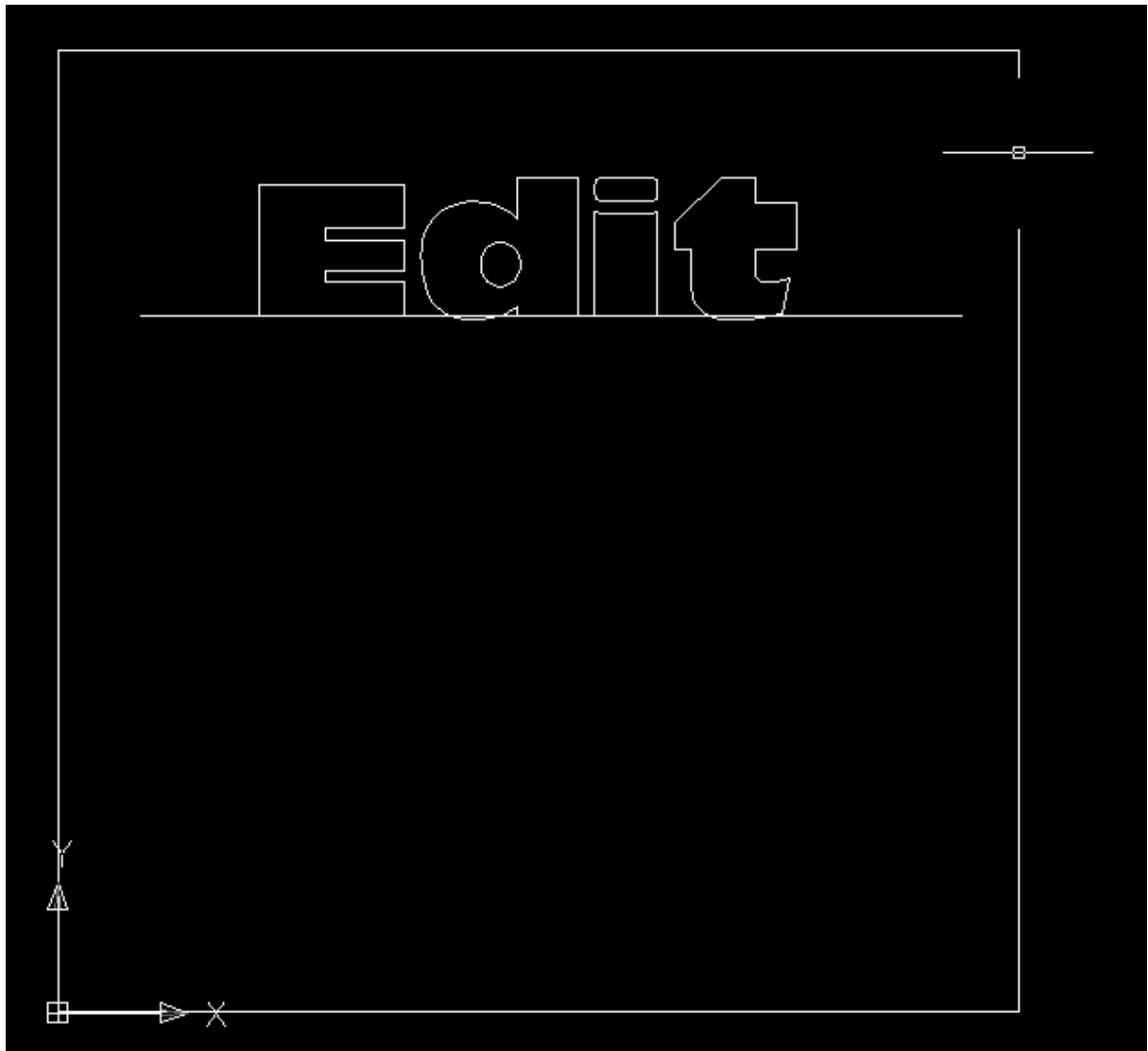
You can now explode and cut the text as necessary.

Edit Text

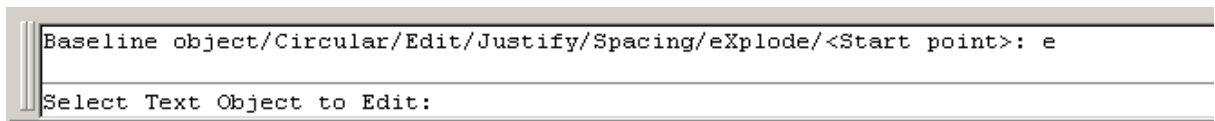
You can use the Edit function in Router-CIM to edit the text you have created on the screen.

This is similar to the edit command in AutoCAD as you will be prompted to select the text on the screen and then you can back up over it and type in new text.

In this drawing the text is already on the screen.



Then select the LE Button and type E for Edit and you will see the following prompt.



Select the text in the drawing and a box will show up at the end of your text line.

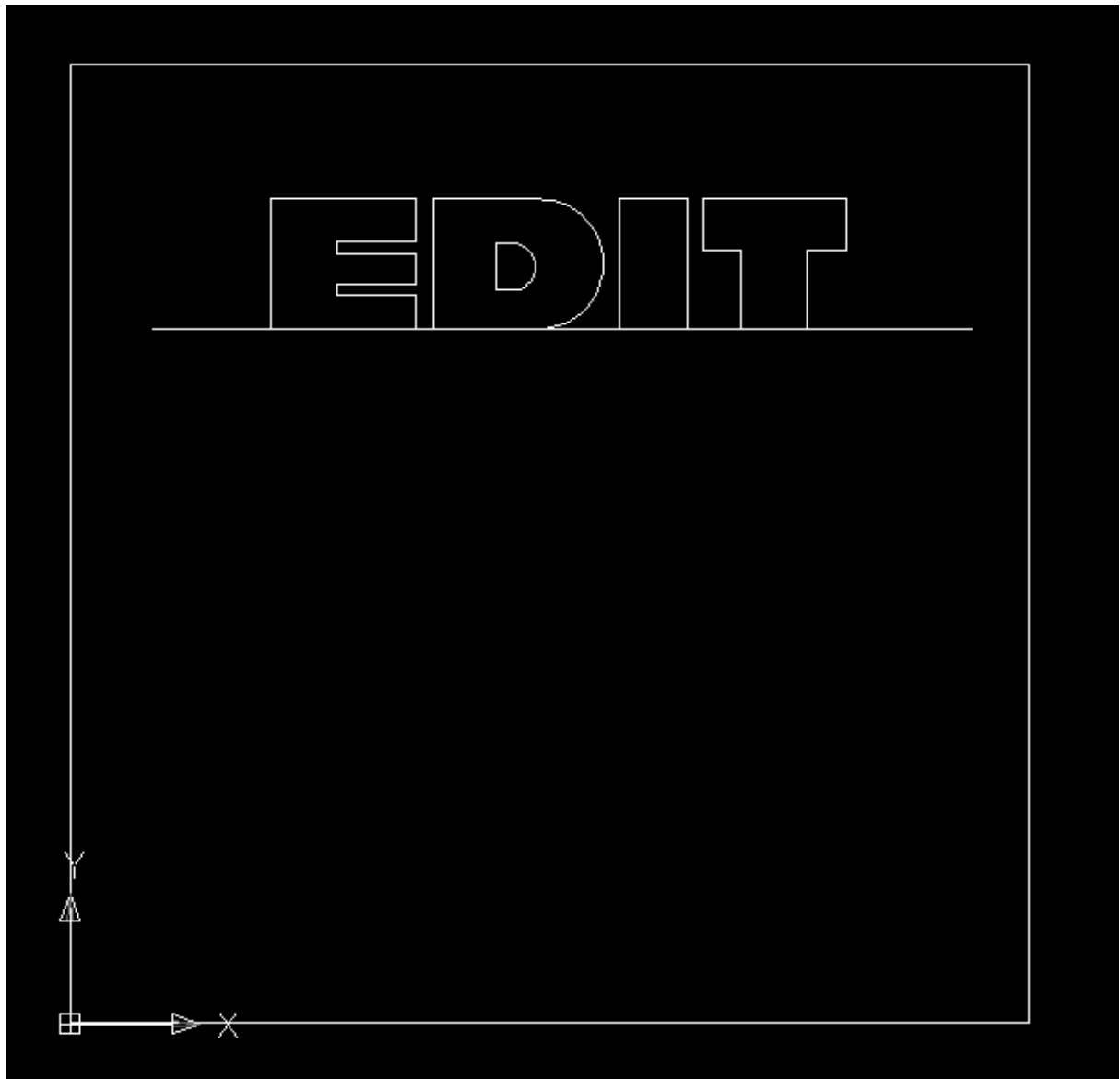


At this point you can backspace over the text and then type in new text.

When finished, you will see the text you typed at the command line and you can press ENTER when you are finished.

```
ENTER-Done, [_EDI<>_]
ENTER-Done, [_EDIT<>_]
```

Your new text should then be on the screen.

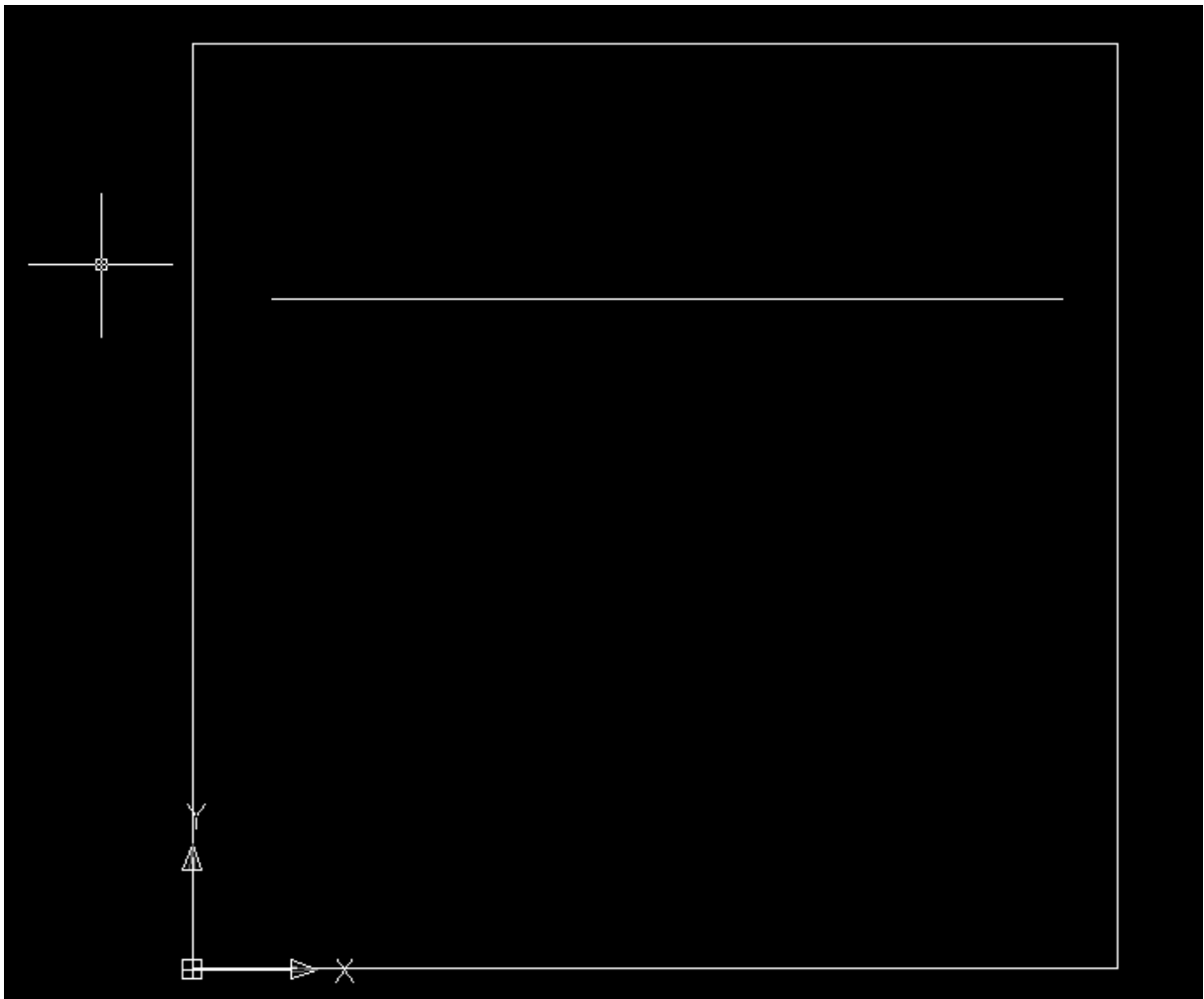


Justify Text

The justify command will allow you to force the text to be aligned in a specific way on your shapes as it is being entered.

You can have text Right Justified, Left Justified, Middle Justified, or spaced to fit in a specific area.

For example, to have text Left Justified to a polyline on the screen, you can enter text in this method.



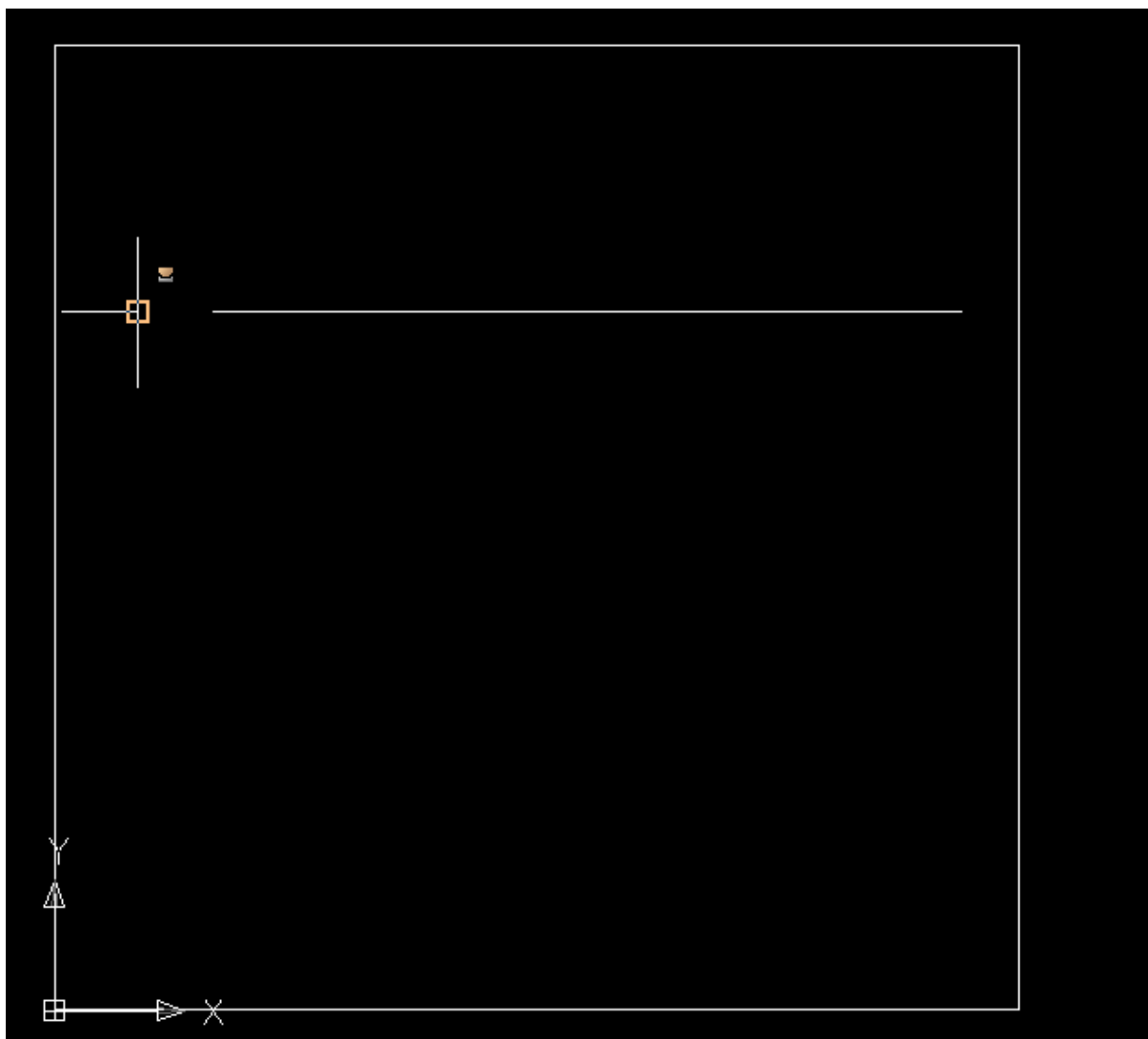
Select the LE Button and at the prompt, select 'J' for Justify. You will see the following prompt at the command line.

```
Baseline object/Circular/Edit/Justify/Spacing/eXplode/<Start point>: j
Enter an option [<Left>/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR/2pt]:
```

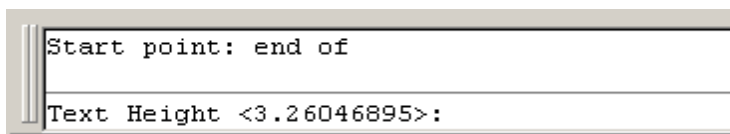
Type 'L' for Left (or just press Enter) and you will be prompted for the Start Point.

```
[<Left>/Center/Right/Align/Middle/Fit/TL/TC/TR/ML/MC/MR/BL/BC/BR/2pt]: l
Start point: |
```

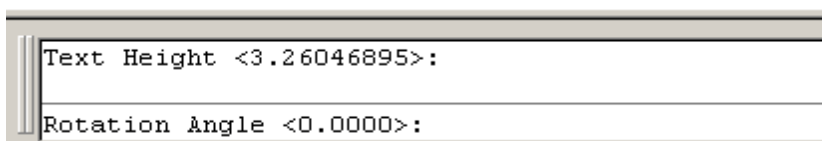
Using your OSNAPS in AutoCAD, select the Left Endpoint of the line.



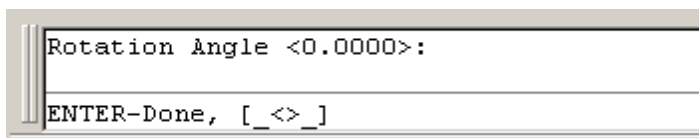
You will then be prompted for the Text Height.



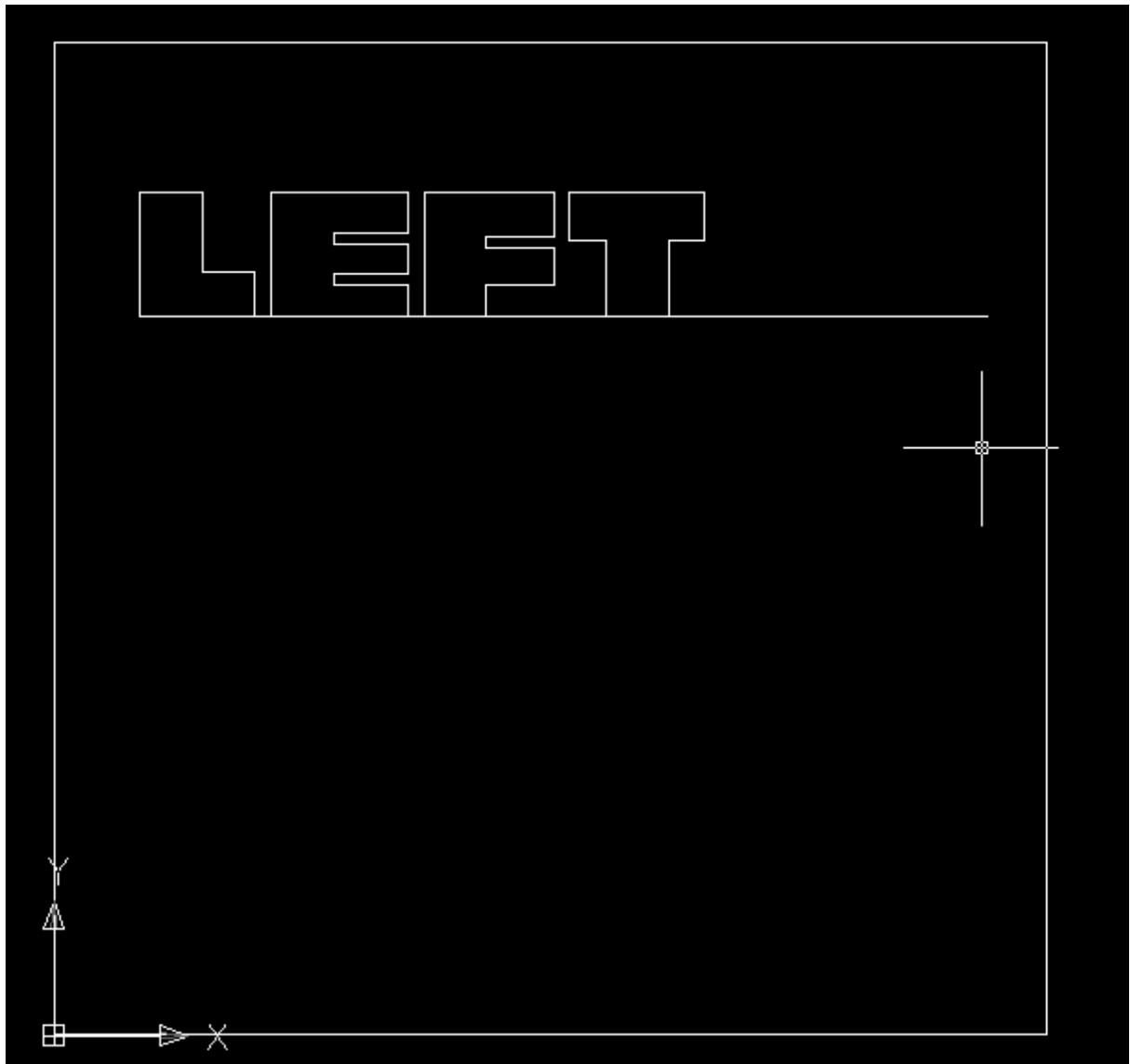
And the Rotation Angle.



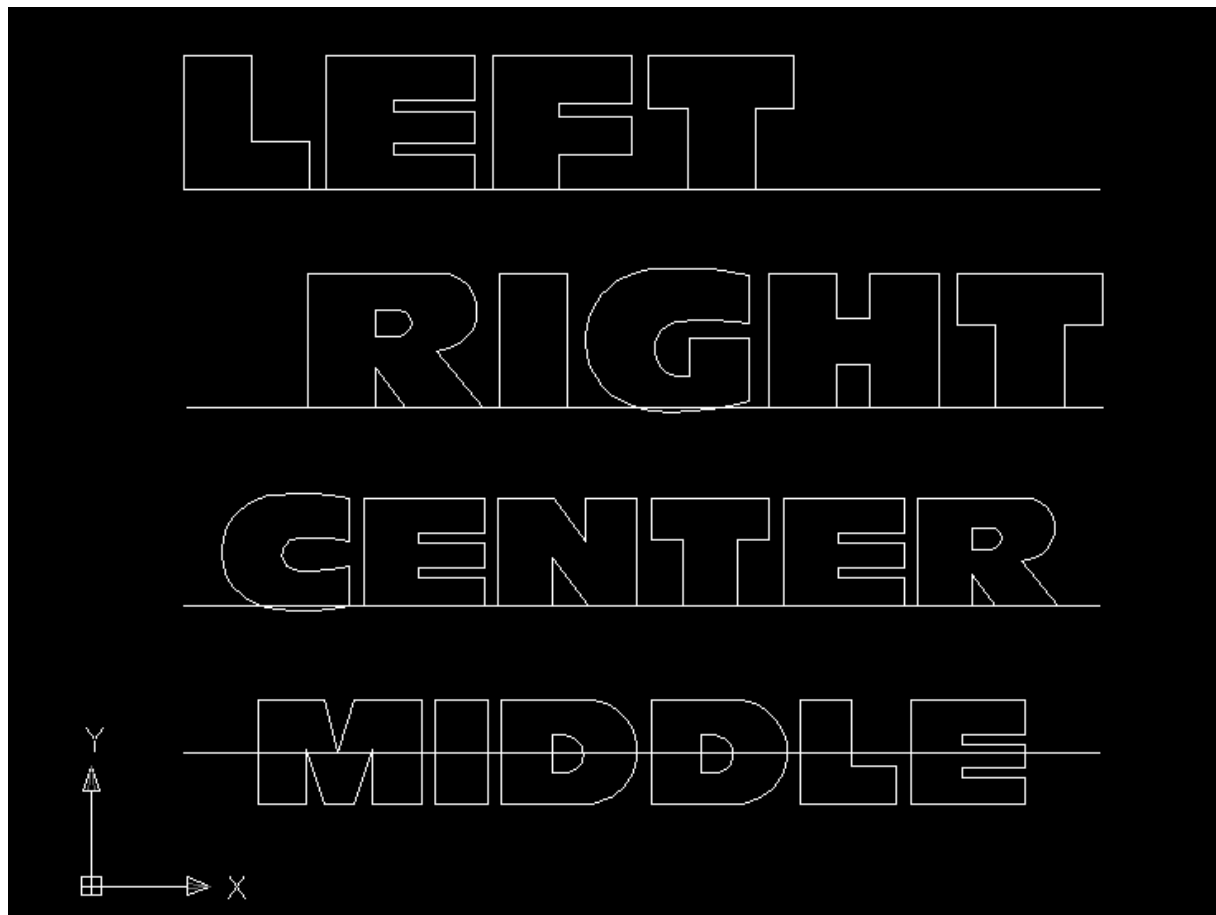
And then you can enter your text.



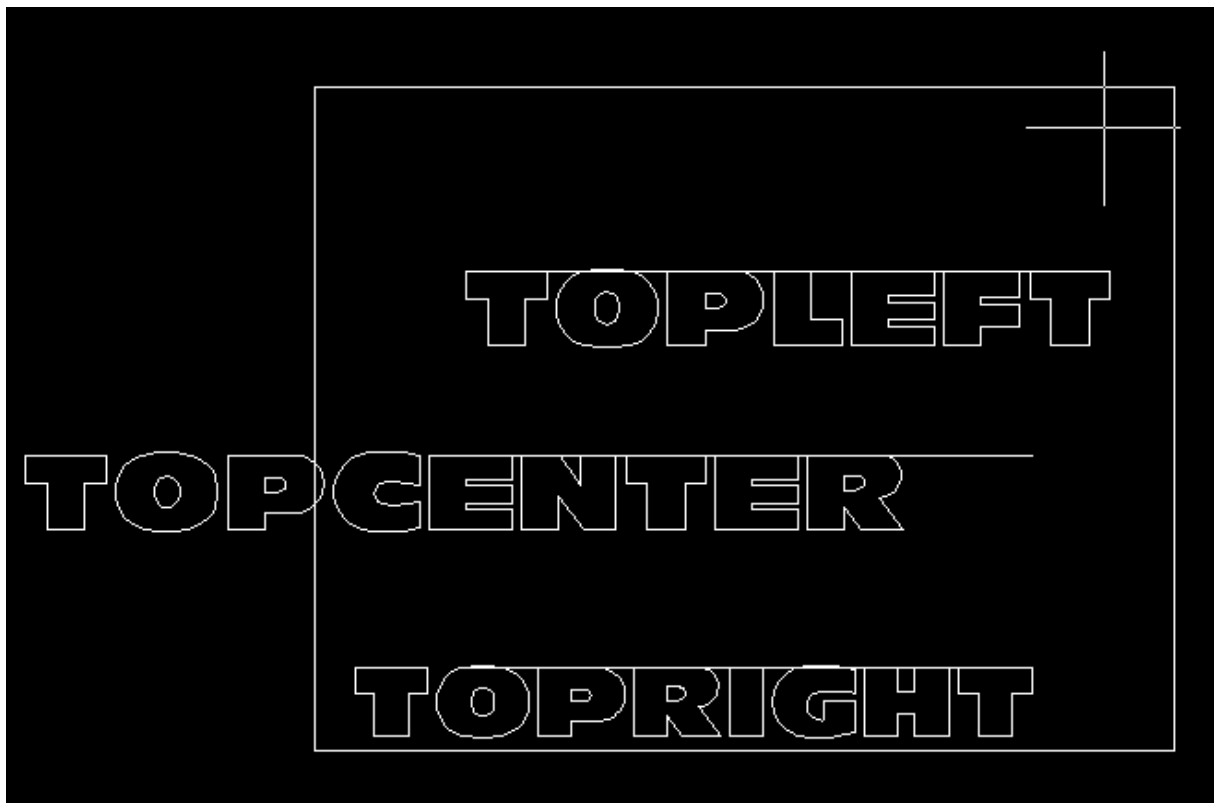
It should then appear to start from exactly the left end of the line where you picked the start point.



There are other prompts for Justify as well.
They are for Right, Center, Middle...



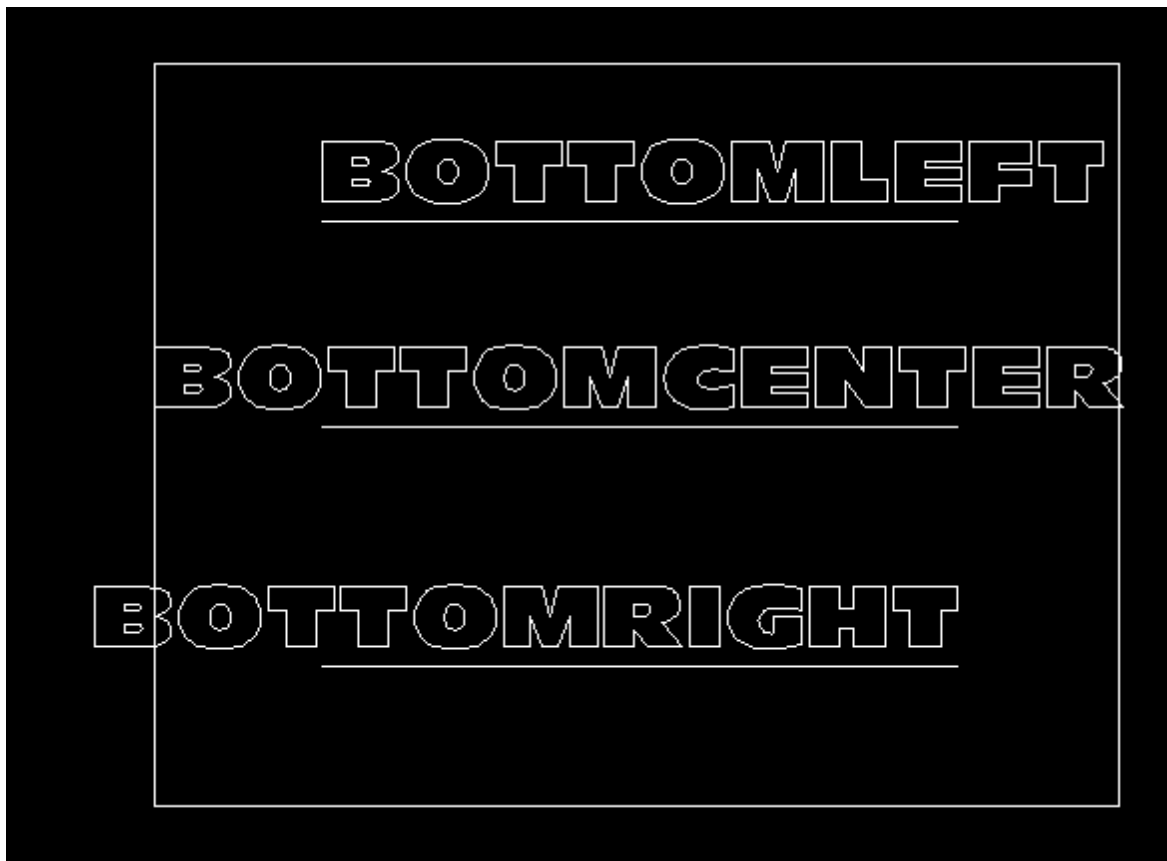
As well as Top Left, Top Center, and Top Right...



And, Middle Left, Middle Right, Middle Center...



And finally, Bottom Left, Bottom Center, and Bottom Right.



There is one final method. That is 2P, or Two Points.

Using this method you would pick two points and the text would be fit between those two points (if possible).

Text Spacing

Spacing is a method used to place blank space between the text to space it out further than normal.

This is a scale factor as it is not in inches or mm's, but is instead a scale of how far the letters are from each other compared to their normal values.

This is not a very common item to change, as it can make the text harder to read, but it is a feature that may at some point be useful.

You can set the spacing to any value from 0.75 - 4.00 as the scale factor in the spacing.

For instance using the normal value of 1.00 and then 2.00 would yield the following result.

SPACING=1.00

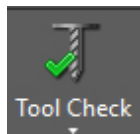
S P A C I N G = 2 . 0 0

Toolcheck

Router-CIM Toolbar:



Router-CIM Ribbon:



Keyboard: TC

There is a function built into Router-CIM to provide a display of the tool along a cut path. This function is called Toolcheck. This display can be used for visual verification and actual measurement checks. To select, click on the Toolcheck icon on the Router-CIM toolbar.

Toolcheck uses only cut cycle blocks as input. A cut cycle block has only one tool description. Toolcheck performs the tooling display using the tool described in a single cut block. If a Sequence group is selected and the Sequence was developed using cycles containing different tools, only the first tool in the Sequence will be displayed. Use Toolcheck on individual Cut cycle blocks for correct tooling display.

The display of the tooling is controlled by four options. These options are Endpoint, Measure, Task, and Emulate. We recommend the use of Endpoint and Measure as the fastest, most reliable means of tool path verification.

The tooling geometry is on the Tool Check layer, and can be erased by selecting the Erasecheck icon off the Router-CIM toolbar. This is the only way to erase a Toolcheck!

Endpoint

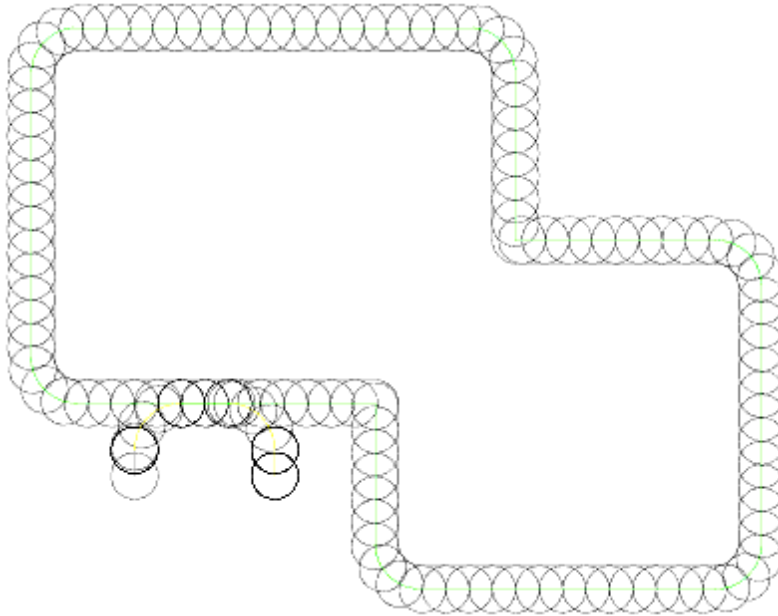
The tool block is inserted at each Endpoint of the tool path. The inserts remain on the screen after completion of Toolcheck.

Measure

The tool block is inserted at Measured points along the tool path. The AutoCAD "Measure" command is used to develop each of the insert points. The spacing of the insert points is entered at the prompt "Enter Tool Check Spacing:". The inserts remain on the screen after completion of Toolcheck. If you wish to use this function to test your tool path, or direction of Cut, etc., click on the Toolcheck icon off the Router-CIM toolbar. You will be prompted to select objects, pick the tool path you wish to have checked. The next prompt gives you specific choices for the Toolcheck mode.

Select 'M' for MEASURE, and you will be prompted for the Toolcheck spacing; usually the default is $\frac{1}{4}$ of the tool diameter. This is the spacing of the Toolcheck circles as they move around the part. To use the default press <Enter>, or change the value, usually setting the spacing to the tool radius is sufficient.

The Toolcheck will then proceed around your part (see diagram below). There will be a circle the same diameter as the tool moving along the center of your tool path at the circles (tools) center point at the spacing you specified.



This is very useful for checking the direction of Cut on open shapes, where it is often difficult to tell if the direction of the Cut is correct.

Task

This option will allow you to use a custom task or macro you design to perform a custom Toolcheck. You will be prompted at the command line with Enter Task Name: and you would type in the name of your task.

A default task would be TC. If you type in TC, the Toolcheck will continue.

Emulate

The Emulate mode is to allow a block representing the tool to move across the tool path at specific intervals.

You will be prompted at the command line for the Tool Block Name. If you have made a custom tool block you want to use, type in the name now, otherwise Router-CIM will insert the block named TDIA. Next, you are prompted to Select Emulation Spacing. The default is usually fine, however, you can set any spacing you desire.

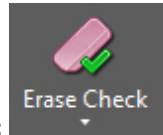
The Starting Element is the next prompt, and you should select the lead-in, then you will be prompted for the Stopping Element, and you should select the lead-out.

Router-CIM will show the tool block moving around the part at the spacing provided.

Erasercheck

Router-CIM Toolbar:





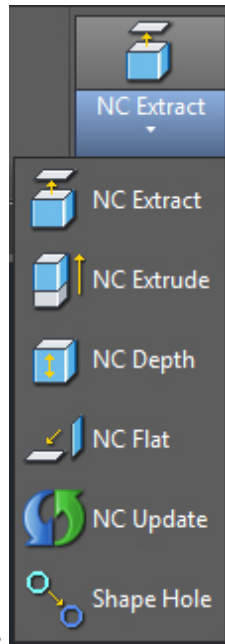
Router-CIM Ribbon:

Keyboard: **ERASECHK**

Use Erasecheck to erase the toolcheck. That is the only way to remove the TOOLCHECK! This can be found on the Router-CIM toolbar, or under the RCIM pull down menu. Please note that ERASECHECK will erase all the circles and then perform a redraw.

Cutting Solids in Router-CIM

Router-CIM Toolbar: 



Router-CIM Ribbon:

The following is taken from the AutoCAD manual to give unfamiliar users some background on Solids.

A solid object represents the entire volume of an object. Solids are the most informational, complete and least ambiguous of the 3D modeling types. Complex solid shapes are also easier to construct and edit than wire frames and meshes.

You create solids from one of the basic solid shapes of box, cone, cylinder, sphere, torus, and wedge or by extruding a 2D object along a path or revolving a 2D object about an axis.

Once you have created a solid in this manner, you can create more complex shapes by combining solids. You can join solids, subtract solids from each other, or find the common volume (overlapping portion) of solids.

Solids can be further modified by filleting, chamfering, or changing the color of their edges. Faces on solids are easily manipulated because they don't require you to draw any new geometry or perform Boolean operations on the solid. AutoCAD also provides commands for slicing a solid into two pieces or obtaining the 2D cross section of a solid.

Like meshes, solids are displayed as wire frames until you hide, shade, or render them.

Additionally, you can analyze solids for their mass properties (volume, moments of inertia, center of gravity, and so on). You can export data about a solid object to applications such as NC (numerical control) milling or FEM (finite element method) analysis. By exploding a solid, you can break it down to mesh and wire frame objects.



- [NCExtrude](#)



- [NCExtract](#)



- [NCDepth](#)



- [NCUpdate](#)



- [NCFlat](#)



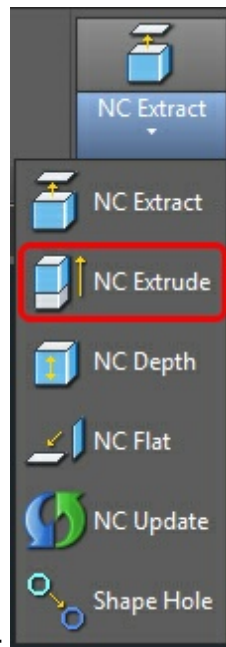
- [Shapehole](#)

You can cut AutoCAD or Mechanical Desktop solid models only if they have been passed through NCExtrude or NCExtract. Do Not Use Geoshape On A Solid!

NCEXTRUDE

Router-CIM Toolbar:





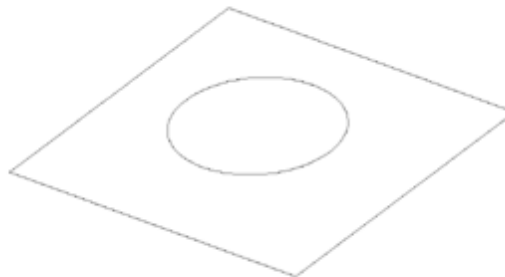
Router-CIM Ribbon:

Keyboard: NCEXTRUDE

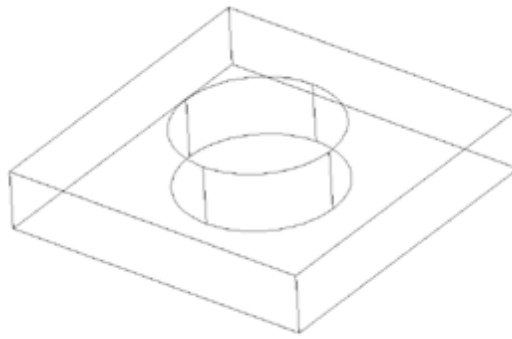
NCEXTRUDE converts a 3DSOLID entity into 2D Polylines that represent the extruded profiles. The thickness of the developed Polylines represent the depth of the profile in the 3D solid. The Polylines are placed on the NC_SHAPE layer and are immediately available for cutting.

All bodies that are in the 3DSOLID that are not extrusions without taper are not converted. 3DSOLID entities that cannot be exploded are not converted.

Example: This will show a simple part being extruded to a 3d Solid and then NCExtrude will sperate the shapes into 2D Polylines with thickness suitable for cutting.



Step1. A simple 2d shape created (shown in an isometric view).

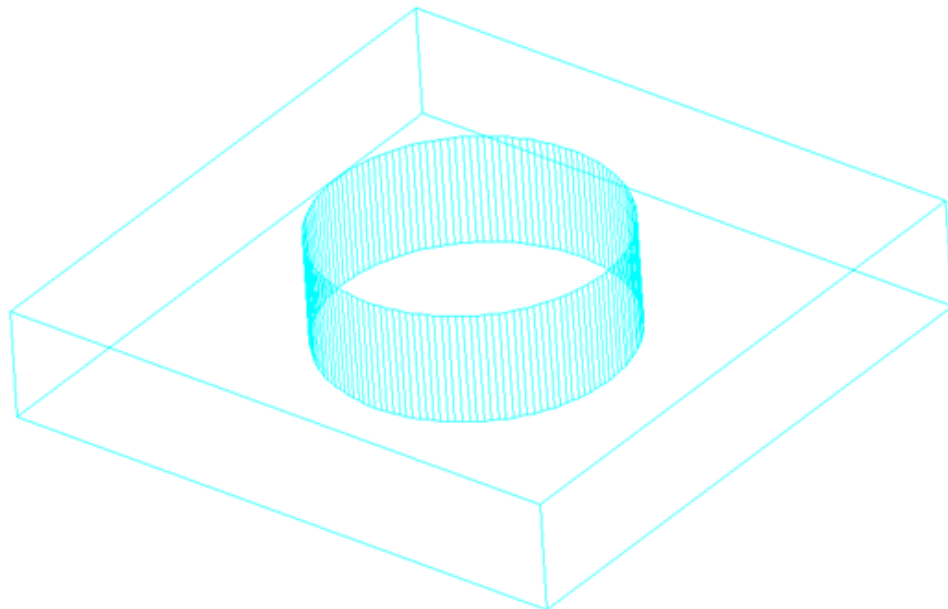


Step2. That same shape extruded and the center shape subtracted from the rectangular shape. This is now 1 solid model.

Step3. The NC EXTRUDE command is run and the following prompts appear:

Initializing...Select Extruded Solid to Define: *Here you select the solid you wish to have extracted*
Examining Selected Solid...
Defining Regions...
Defining Shapes...
Defining Shape(s)...
Defining Shape(s)...

The resulting shape looks like this:

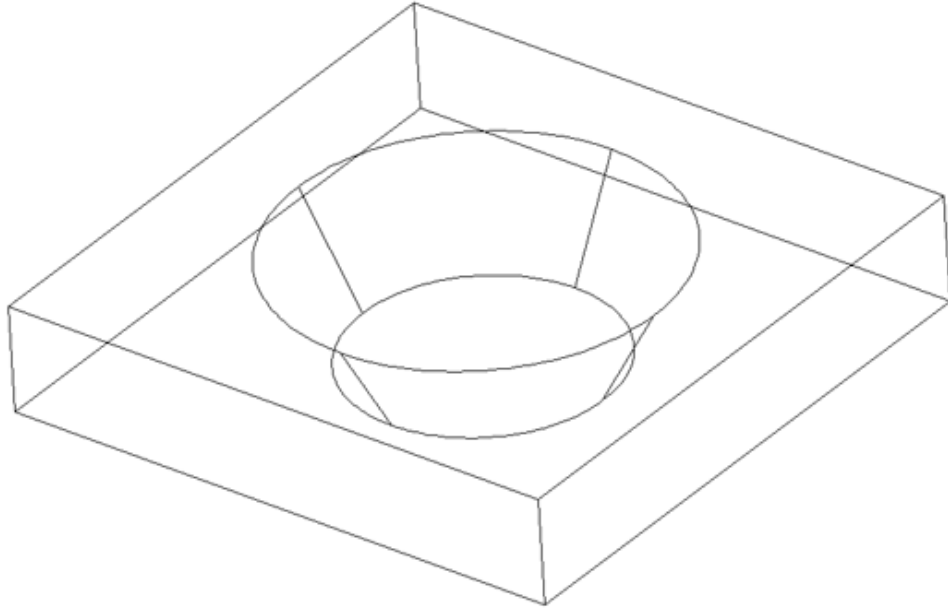


These resulting shapes are now reading for cutting in Router-CIM. If you wish to cut them with a layer to knowledge association, they resulting geometry shown here will have to be placed on the layers you wish to use for the layer to knowledge association. NCExtrude will only place the geometry on layer NC_Shape in preparation for cutting.

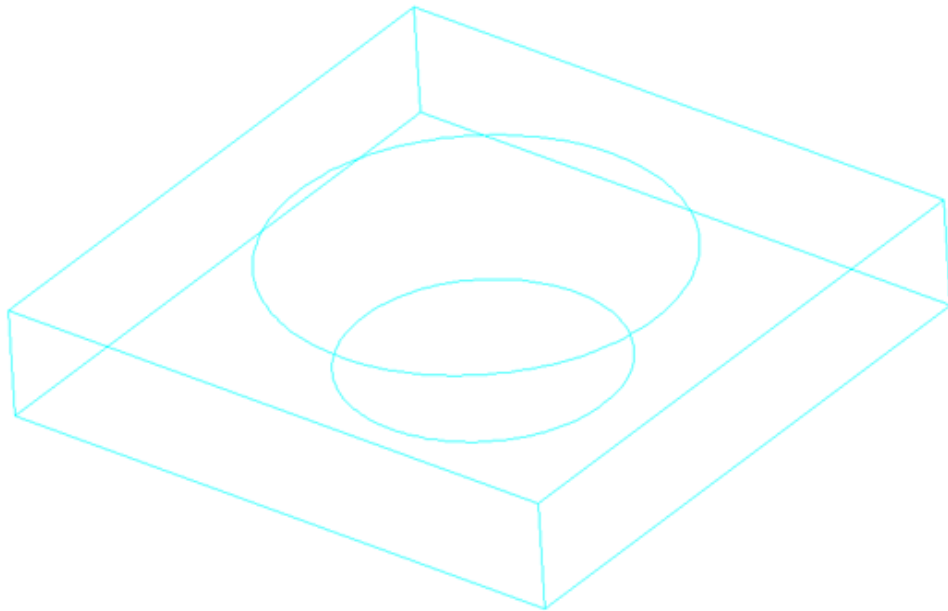
Objects with Taper

Only shapes that have been extruded and not tapered will have thickness assigned to the resulting 2D polyline created with the NCExtrude command.

If an object has a taper to it, NCExtrude will provide a 2D shape on the top and the bottom of the shape. Both will be on layer NC_Shape and suitable for cutting, but neither will have thickness.



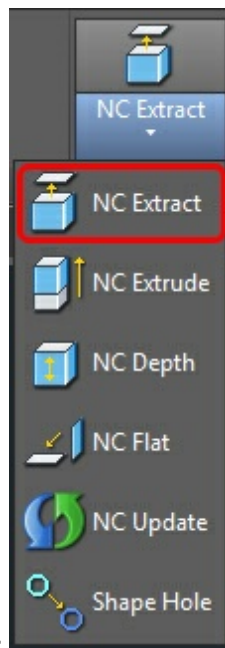
This center shape clearly has a taper to it.



The resulting shape from NCExtrude is a 2D profile with thickness on the outside shape and two 2D profiles on the inside shape. One at the top of the shape and one at the bottom.

NCEXTRACT

Router-CIM Toolbar:



Router-CIM Ribbon:

Keyboard: **NCEXTRACT**

Can be used on an AutoCAD or Mechanical Desktop Solid. You get a Geoshaped polyline from a planar face.

The NCEXTRACT command creates Shape Polylines from Solid models. NCEXTRACT works on core solids and parametric solid models.

Using NCEXTRACT:

From the RCIM pull down menu, pick Define Part >> Define Solids >> NCextract.

The NCEXTRACT command will extract a Geoshaped polyline from the face of an AutoCAD solid or Mechanical Desktop solid. That polyline is ready for start point edit or cut. By using this command, you create a NC friendly shape to be cut. This shape can be moved or the start point changed. This shape can also be used by the system to determine the positioning information for NCRotate and alignment of the cut.

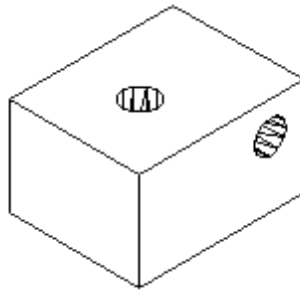
Start the NCEXTRACT command from the toolbar, menu or the command line. The system prompts you to select an edge of the part. After selecting the edge, the system will highlight on of the two

faces connected to that edge and ask you to accept this face. If it is the face you want, type **Y** for yes, otherwise type **N** for no or hit enter to accept the default. If you enter **No**, the other face that is attached to the edge will highlight, and the question will repeat.

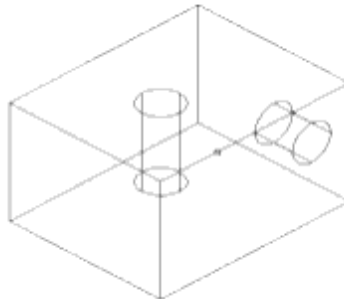
When you accept a face, all of the edges on that face will be converted to shape Polylines automatically. You may then start point edit, move or cut these shapes.

Example:

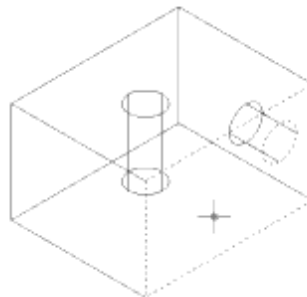
We would like to extract shapes on the top of this solid model:



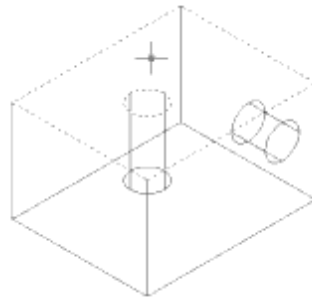
To extract from a Face, you pick the solid on an edge as shown:



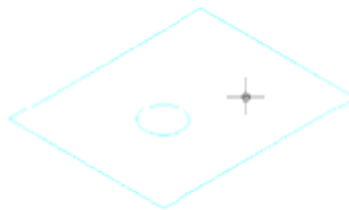
The first of the two faces will highlight. This is not the face we want, so we hit N on the Accept prompt.



This is the face we want, so we hit Y at the Accept prompt.



These are the shapes created. The solid has been removed for clarity.



When using NCEXTRACT, pick the face to extract and either Accept the selected face or toggle to the Next face. This face is automatically Geoshaped. You still need to use Start point Edit to choose where to lead in and out.

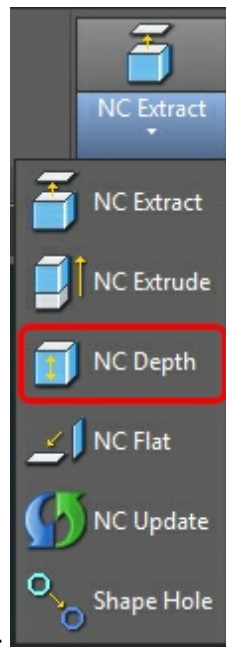
When using NCEXTRACT with any Mechanical Desktop solid, the resulting NC_shape polyline is parametrically attached to the solid. This attachment is two-dimensional. Whenever the solid changes size, the NCUPDATE command will update the NC_shape polyline. The NCDEPTH command makes the attachment three-dimensional. NCDEPTH is only necessary if you want the polyline to maintain a thickness associated with a solid. NCDEPTH is especially useful when using "A" for the total depth.

Known issues using NCEXTRACT: If you change the start point on a shape that has been extracted, the start point will revert back to an arbitrary location after NCUPDATE.

NCDepth

Router-CIM Toolbar:



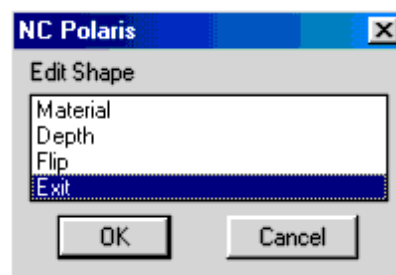


Router-CIM Ribbon:

Keyboard: NCDEPTH

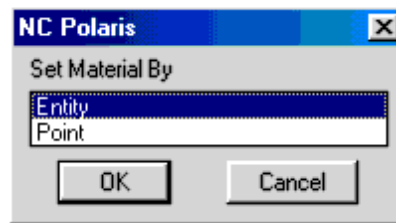
The NCDEPTH command allows the user to take existing shapes and link the material location and cut depth to existing entities, points in the drawing or specific values. The material location as well as the depth of the cut are defined using this command. For graphic feedback, the resultant shapes are moved up or down relative to the position of the polyline for the material location. The thickness of the shape is updated to show the depth of the cut. This depth of cut can be overridden by inserting a depth of cut in the Knowledge Editor. The cut depth can also be adjusted by a value. Use the / character before the number in the Knowledge Editor to add or remove from the depth of defined by NCDEPTH.

The NCDEPTH command has several options:



Material

The material location for a cut is defined by the location of the polyline. When you select Material, you are prompted for the shapes to edit. Select as many shapes as you wish. All shapes will be linked to the same material location. When you have selected the shapes, decide how to define the material location.

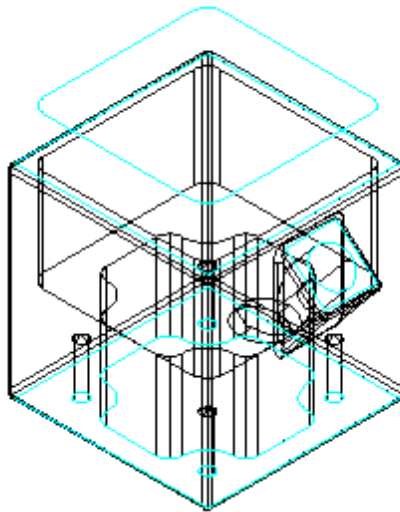


The ENTITY option prompts for an entity. Any entity that defines a point can be used to define the material location for a shape. If a 3D Solid object is selected, you will be asked to define the specific edge of the solid that should be used to define the material location.

The POINT option defines a specific point in WCS that defines the material location.

Regardless of how it is defined, by ENTITY or by POINT, the point does not change the work plane defined by the polyline. The polyline plane will be moved up or down to match the location of the defining point. Careful consideration should be made when cutting on the 4th and 5th rotational axis for proper positioning of the shapes.

For example, this part shows a shape polyline that defines the pocket geometry of the solid model to cut. The part model was moved down after the shape was created. The shape is now too far above the part to be used. The shape location should follow the top edge of the solid model. The material option of the NCDEPTH command allows you to link the material location of the shape polyline to the geometry of the solid model.



When you choose the Material Option, by Entity, the first prompt asks:

Select the Shapes to Define New Material Location:

Select objects: 1 found

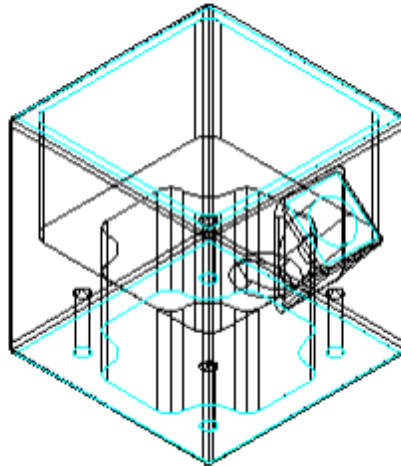
The shape above the part is selected. Choose the Entity option and select the solid model.

Select Material Object:

Solid Selected!

Select Edge on Solid

Since this is a solid model, you cannot select a specific edge until the solid is selected. The solid model is highlighted at this time, so select the top edge of the pocket hole.

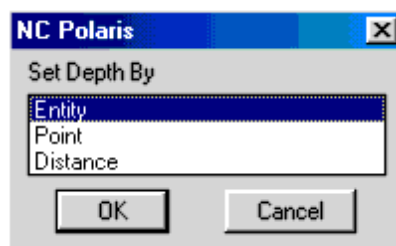


The shape moves down to the top of the hole.

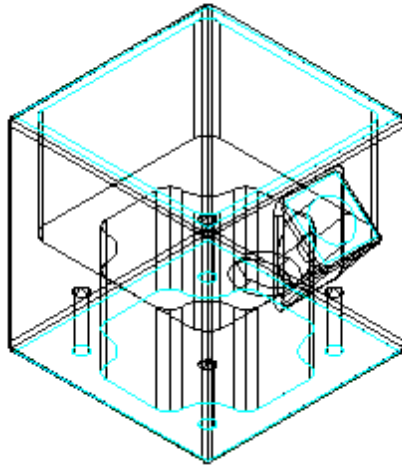
Depth

The depth of a cut is defined as the thickness of the polyline. The thickness is used by Router-CIM when you leave the Total Cut Depth blank, or fill in a plus or minus value that has a '/' <forward slash> in front of it. When blank, the exact thickness of the shape is used. When a Slash Value is used, it is added or subtracted from the actual part thickness. This can be used as an over cutting or undercutting technique.

In our example, the top shape is in position for the material location, and now we must define the cut depth. Cut depth can be defined in three ways:



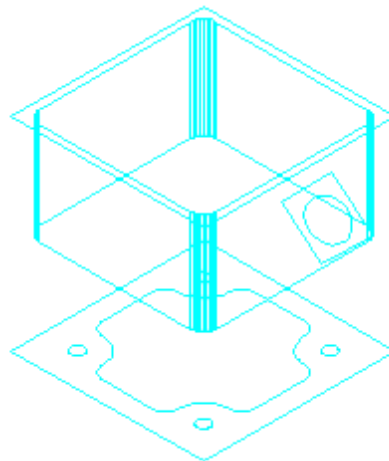
Entity and Point work just like defining the Material location. Distance prompts for a simple depth. For example:



Select the top pocket shape
Select the Shapes to Define New Depth:
Select objects: 1 found

Select the Entity Option, then select the Solid.
Select Depth Object:
Solid Selected!
Select Edge on Solid

The solid edge selected is an edge on the bottom of the pocket shape. The polyline is automatically updated to show the new shape depth. The solid has been removed from this picture for clarity.



Flip

This option allows the user to flip the cutting direction of the polyline to the opposite side. Some shapes are easier to extract from the bottom or back side of a model and need to be flipped to cut properly.

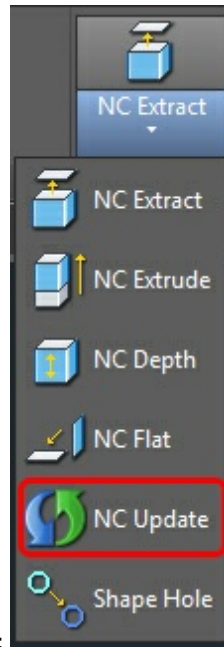
Select the Shapes to Flip:

Select objects:

The selected Polylines will be flipped.

NCUpdate

Router-CIM Toolbar: 



Router-CIM Ribbon:

Keyboard: **NCUPDATE**

The NCUPDATE command recuts multiple cut blocks. A whole sequence can be updated to new geometry. Geometry that has links to other geometry will be updated as well. The NCEXTRACT command links geometry to the face of the solid object it came from. The NCDEPTH command can link shapes to objects that define the material location and cut depth. The STARTPT command can change the start point of the cut. The Move and Rotate commands can change the location and orientation of the shape. The NCUPDATE command searches for the changed shapes and updates them before updating the cuts. If a sequence of cuts is selected, the NCUPDATE command launches the NC Sequence Builder dialog box to allow the user to redefine the sequence, change sorting options, make new NC code, and generate new reports from the cut blocks.

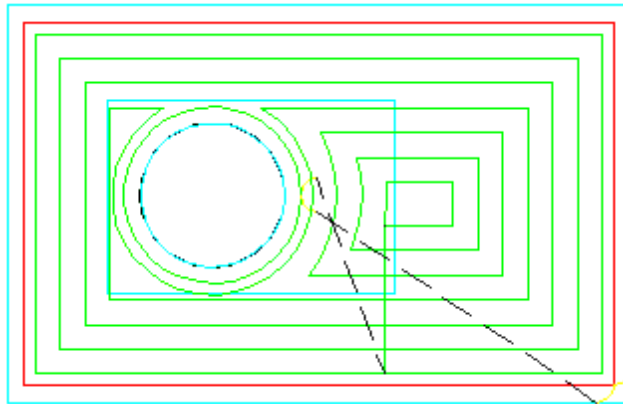
The only input for NCUPDATE is:

Select Cuts to Update:

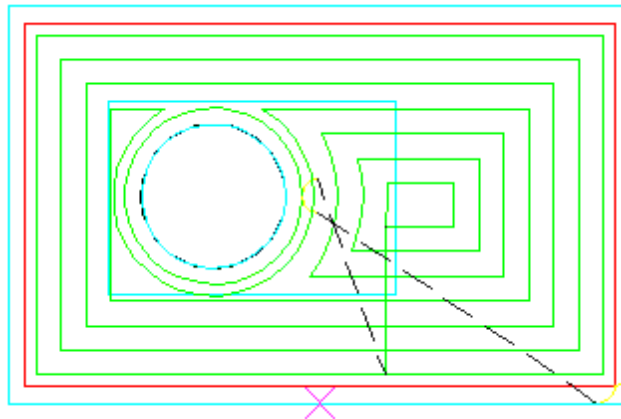
Select Objects:

Select the cut blocks to update. The cuts will be recreated on the screen. Any changes in the source geometry will affect the results of the new cut.

For example, this simple pocket has a new island shape defined for it:

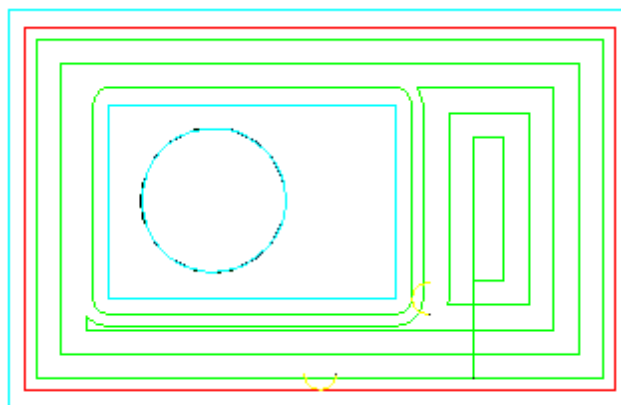


The rectangle is the new island shape. The circle is linked to the pocket and contour cut. Also, note that the start point of the contour cut on the large rectangle is in the wrong place. First, we use the Start Point Edit command to modify the outside shape start point to a better position.

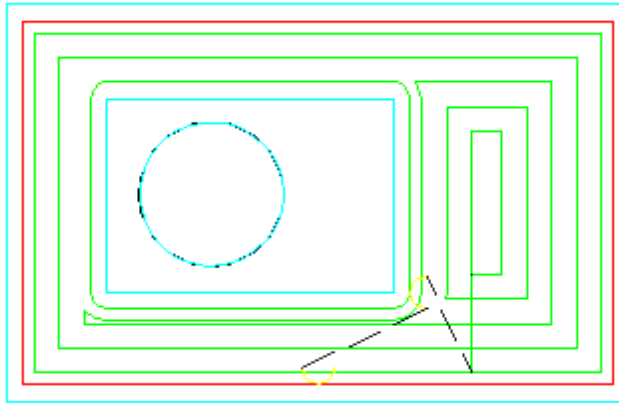


Then we use the NCLINKCUT command to link new shapes to the cut. Since NCLINKCUT only works on one cut block at a time, we select the pocket cut first. It is linked to the outer rectangle and the inner rectangle. The contour cut is only linked to the inner rectangle.

Now that the cuts are linked to the new shapes, the NCUPDATE command is run. The sequence of cuts is selected when you choose any one of the cuts, and the cuts are automatically updated to the new geometry.

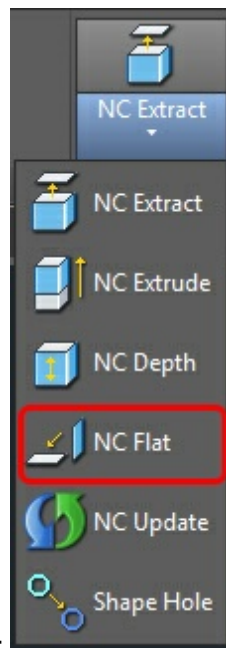


The index lines are removed from between the cuts, and the sequence options dialog box is displayed. The options that were used to create the original sequence are displayed. They can be edited if you desire, or hit OK to recreate the sequence as is. New index motions are created between the cuts.



NCFLAT

Router-CIM Toolbar:



Router-CIM Ribbon:

Keyboard: **NCFLAT**

This icon is on the RCIM Solids toolbar. You may also access this function at the command line by typing `ncflat`. NCFLAT will rotate a part in space so that a selected face is pointing up to AutoCAD WCS (World Coordinate System). The part will also be extracted as if NCEXTRACT had been run.

This command is used only on Solids and Assemblies in AutoCAD or Mechanical Desktop. If you wish to use NCFLAT on a part of an assembly, it is recommended you copy the part and run the command on the copy. This allows you to use the Update Assembly command in Mechanical Desktop and have any changes to features, constraints, or dimensions reflected in the update.

To run the command:

1. Click on the icon and you will be prompted:

Select Face

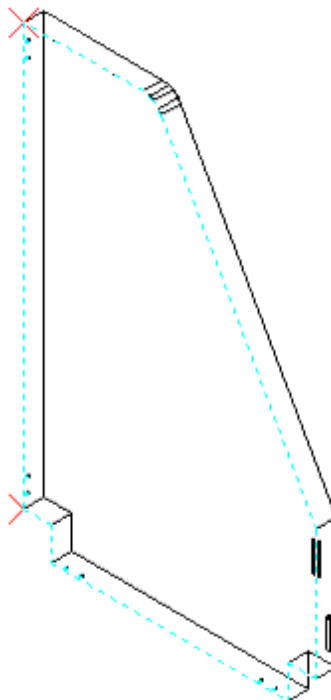
Select the face of the part to orient up. Depending on the part, you may need to choose Next to select the proper face. The selected face will highlight. Hit Enter once you have the proper face selected.

2. The face will appear as a blue dashed line with red Xs appearing at the endpoints of the longest segment. At the command line you will be prompted:

Pick origin point <Longside>

Running object snaps will be changed to Endpoint for the next two prompts.

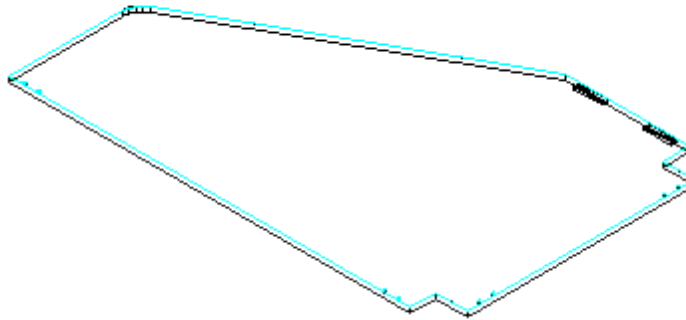
3. Choose the origin base point or hit Enter for automatic longest side selection. The longest side will be noted with a small X at the base point and a small > (arrow) at the x axis position.



If you chose a origin point, select a point that defines where the Positive X-axis should be.

4. Then choose the move to point. It defaults to AutoCAD World 0,0,0. Running object snaps will be turned off for this pick. A drag view of the part will be shown. World 0,0,0 is the base point of the dragging.

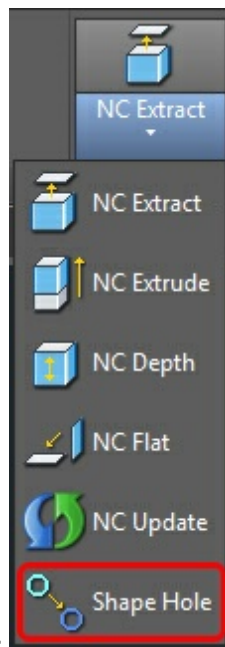
The part will be moved to the move to point at the appropriate orientation in three-d space. The part will be extracted just as if NC Extract had been run.



Note: The longest side calculation only examines the ends of the outside profile. If the longest feature on the outside of the part is an arc, the ends of the arc will be lined up in the X-axis.

Shapehole

Router-CIM Toolbar:



Router-CIM Ribbon:

Keyboard: **SHAPEHOLE**

Shapehole is a command that was developed to allow the result of the ncextract geometry to be turned back into holes for Pattern Recognition in Router-CIM.

After an NCExtract or NCExtrude has been performed, and even after NCFlat has been run to put the part in the world coordinate system, use the Shapehole command and select all the circle shapes that

you want to be recognized. The result will be circles placed on Layer Drilling to be sent to Pattern Recognition.

If you want to use Drill motions manually, then this step is not necessary.

Parametric Toolpath in Router-CIM

Parametric Tool Path capability in Router-CIM and Mechanical Desktop.

Parametric tool paths work on Router-CIM 2D toolpaths and extruded solids in MCAD. The solid must start at Z0 and be extruded in the negative Z direction. To use this feature, follow these steps:

- 1) Draw, constrain, and extrude a shape to create a Mechanical Desktop Solid.
- 2) Load Router-CIM.
- 3) Use the NCEXTRACT command and select an edge on the solid that is to be cut. This command will make an NC_SHAPE polyline on the solid. Typing N for Next and A for Accept can toggle each edge.
- 4) Type NCDEPTH to associate the shape with the parametric solid.
- 5) Pick DEPTH, and at the prompt: Select the Shapes to Define New Depth: select the NC_shape polyline that was created by Ncextract, and then hit Enter.
- 6) Pick ENTITY, and at the prompt: Select Depth Object: select the bottom of the solid. At the prompt: Select Edge on Solid, pick the bottom edge of the solid again.
- 7) Pick OK to EXIT. The result of the NCDEPTH command is an NC_shape polyline that has the same thickness as the solid.
- 8) To make a tool path, pick a tool and cycle, leave the Total Cut Depth field blank, and CUT the shape that was created by Ncdepth.
- 9) You can edit the solid by double picking on the Extrusion in the Browser. When you change the dimensions, the solid will update and get bigger or smaller.
- 10) Type or pick NCUPDATE, and select the tool path. This will update the shape and tool path to the newly sized solid. If the solid gets thicker, Depth per pass will start to make multiple Z level tool paths.

The same is true of a standard 2D polyline that was Geoshaped and cut in Router-CIM. Stretching the shape and using the NCUpdate command will change the toolpath.

Cutting an Ellipse

Sometimes cutting an ellipse in AutoCAD can be a tricky event. You draw out the ellipse you want, and then use Router-CIM to cut it and determine later that the ellipse shape is not really smooth or not the shape you drew at all.

The problem that usually occurs is that in AutoCAD, PELLIPSE is set to 0, and the geometry actually lists as an ELLIPSE. But, in Router-CIM, ***NOPELLIPSE*** is set to *nil*, so when the ellipse is geoshaped, Router-CIM turns **on** PELLIPSE and uses AutoCAD to draw a Polyline ELLIPSE, which is not really an ellipse, but an approximation of one drawn as a polyline. Then the two pieces do not match and you end up cutting a shape that is not the same as the one you drew.

If you have an Ellipse that does not seem to cut right...

- Check that the Geoshape follows the original geometry and does not deviate or 'wander' in and out of tolerance.
- List the original geometry. See if it is an ellipse or a polyline.
- Check the PELLIPSE setting in AutoCAD
- Check the *NOPELLIPSE* setting in Router-CIM

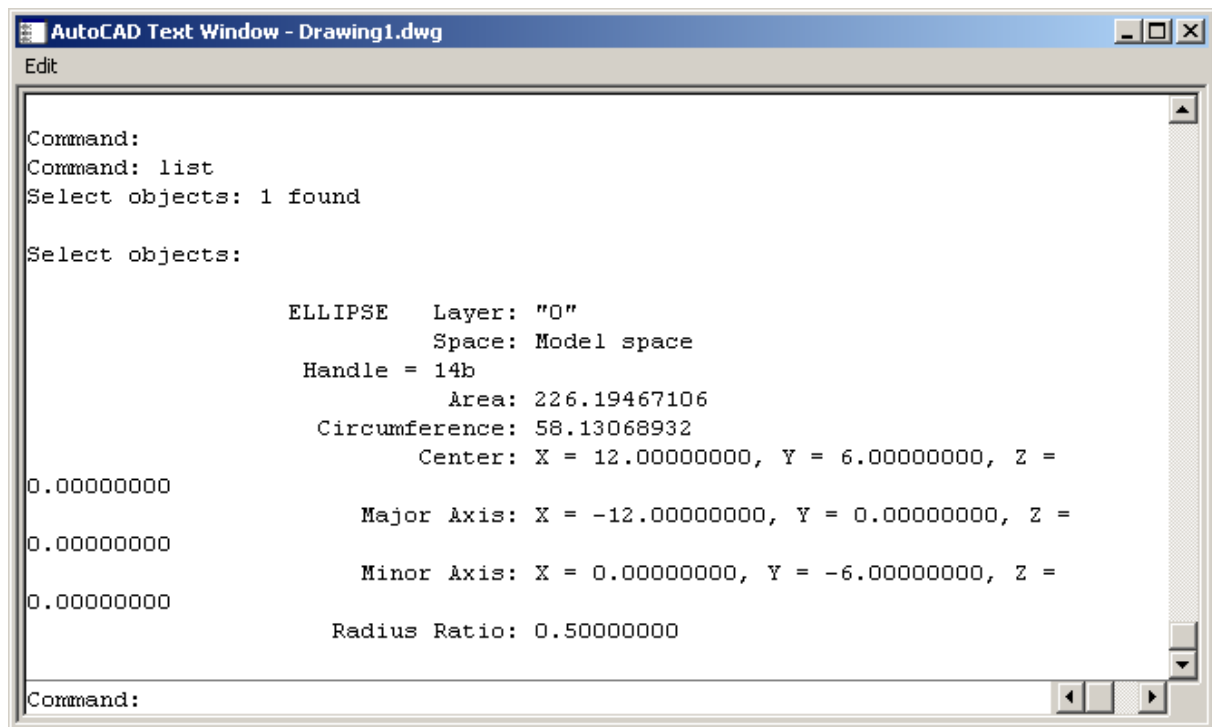
If the AutoCAD geometry is an ELLIPSE when you list it (see below) then you must set ***NOPELLIPSE*** to T in Router-CIM in the NCVARS.

If the AutoCAD geometry is a POLYLINE when you list it then you should set ***NOPELLIPSE*** to nil in Router-CIM in the NCVARS.

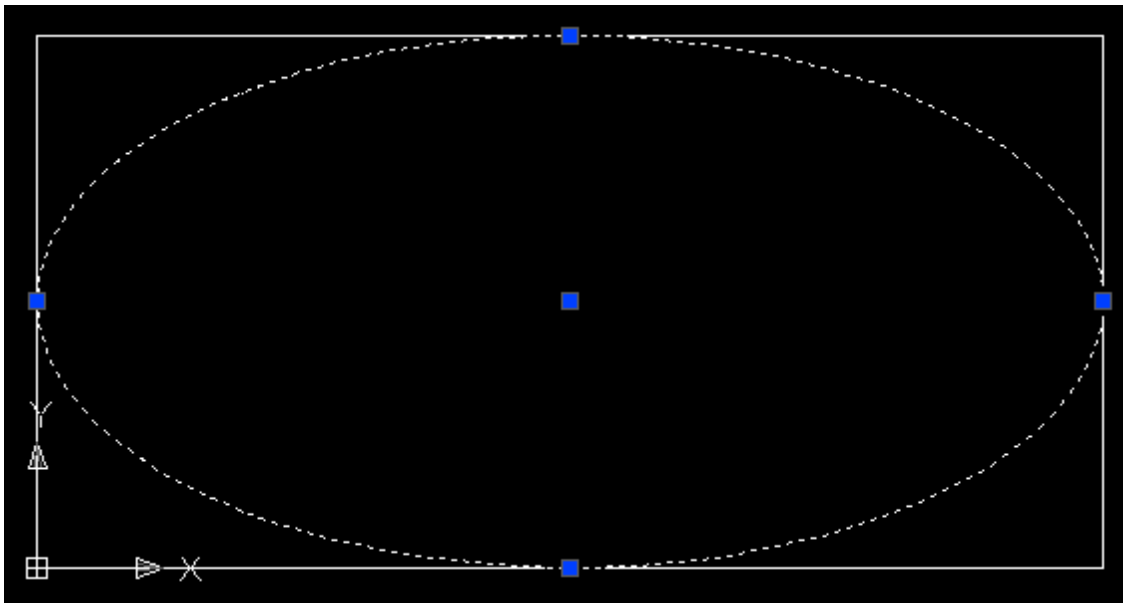
Please refer to the ['NCVar Editor'](#) section for information on how to add an NCVAR.

Listing the object in AutoCAD:

Once you have drawn your shape, type LIST at the command line and at the Select Objects prompt pick on your shape and hit ENTER. AutoCAD will show you the data for the object. The listing below shows a listing for an ellipse with the PELLIPSE variable set to 0.



If you select the object in AutoCAD, you will only see grips for the center and endpoints of the Major/Minor Axes.



This more or less demonstrates that the ellipse is really an ellipse and not a polyline.

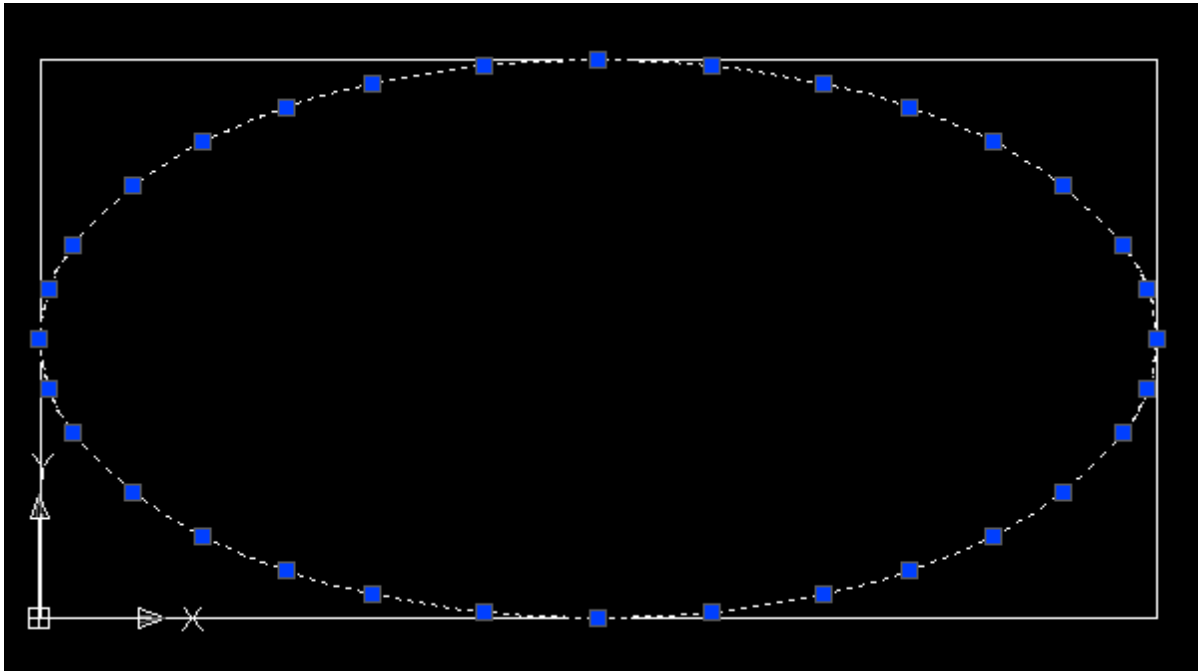
If you set PELLIPSE to 1 in AutoCAD (type PELLIPSE at the command prompt and hit ENTER, when prompted for a value, 1 will make ellipses as Polylines, and 0 will make the as actual ellipses), and then draw the same ellipse again, and list it, you will see this:

```
AutoCAD Text Window - Drawing1.dwg
Edit
Select objects:

      POLYLINE   Layer: "0"
                  Space: Model space
                  Handle = 63d
                  Closed
starting width 0.00000000
ending width 0.00000000
      area 226.09207119
perimeter 58.10161303

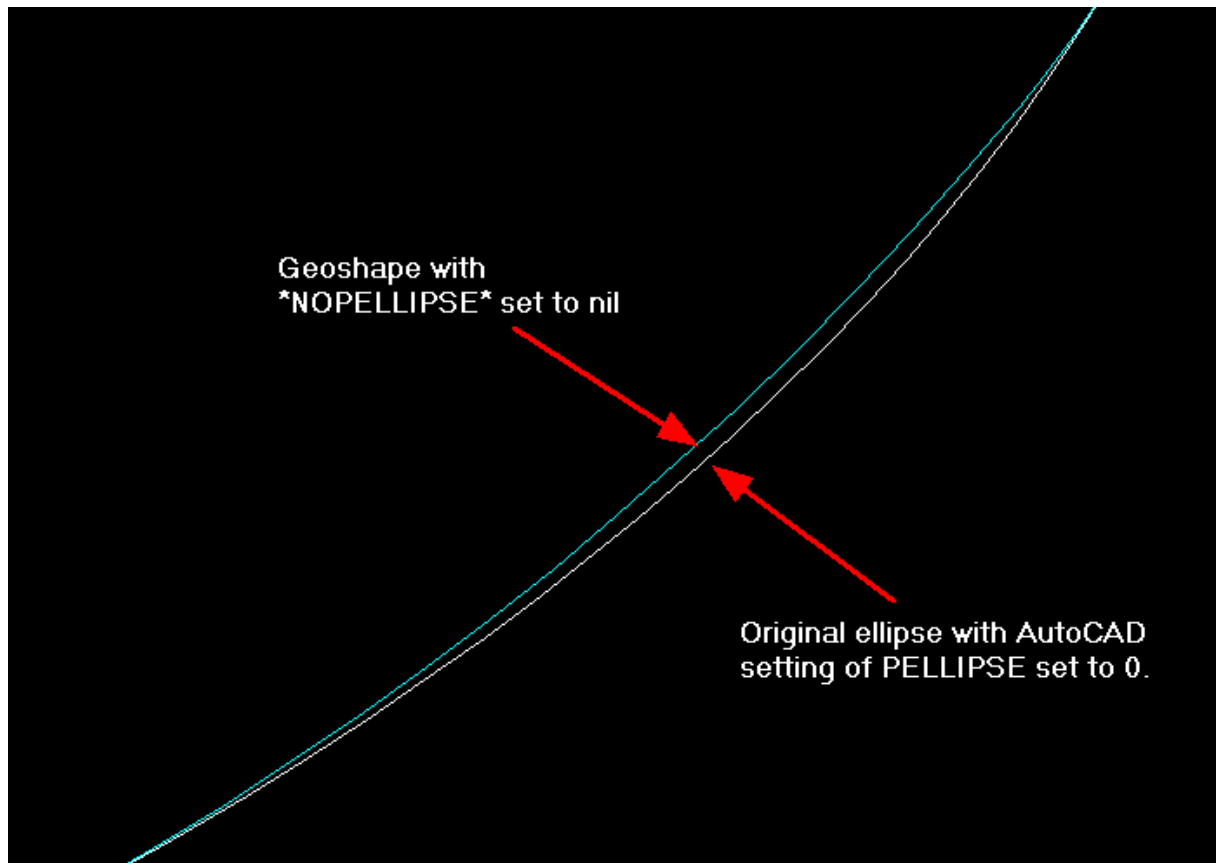
      VERTEX     Layer: "0"
                  Space: Model space
                  Handle = 63e
                  at point, X=0.00000000 Y=6.00000000 Z=0.00000000
starting width 0.00000000
ending width 0.00000000
      bulge 0.17470452
      center X=3.18604281 Y=6.00000000 Z=0.00000000
Press ENTER to continue:
```

Showing you that the ellipse is now a Polyline. If you select it on the screen, you will see many more grips as the ellipse is made up from 16 arc segments, which is the absolute minimum number of arcs you can use to approximate an ellipse (which is what causes all the trouble as there really should be at least 4 in each quadrant, which means 32 total or twice the AutoCAD default resolution).

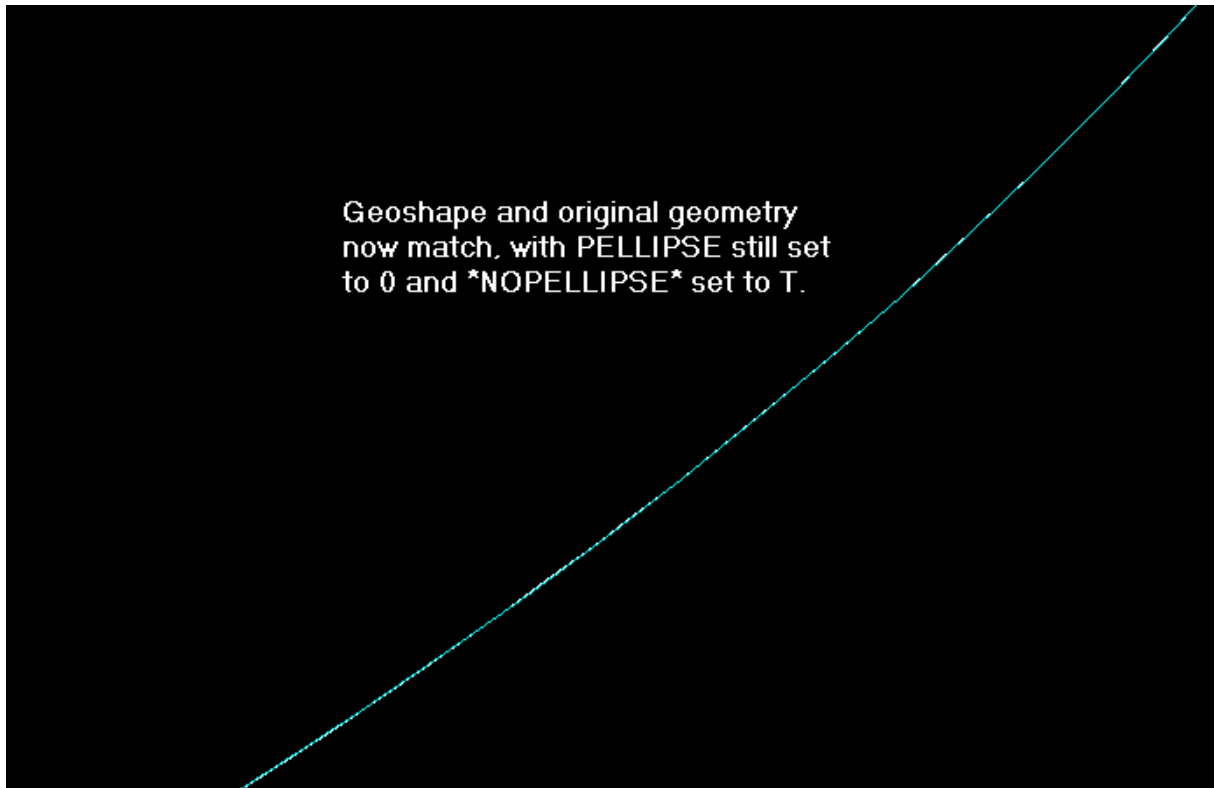


Now that you know how to determine if your geometry is an ellipse or a Polyline, you have to know how to deal with it in Router-CIM.

The trouble occurs when you draw a regular ellipse in AutoCAD (PELLIPSE is set to 0 and you only see a few grips like the first example) and in Router-CIM the *NOPELLIPSE* variable is set to nil. That means that Router-CIM will USE the PELLIPSE routine when it creates its geoshape and approximate the ellipse in 16 arcs. The two elements will not match because the real ellipse geometry is created with more resolution than 16 arcs can approximate. When you geoshape then you will see this type of 'wandering' on the geoshape:

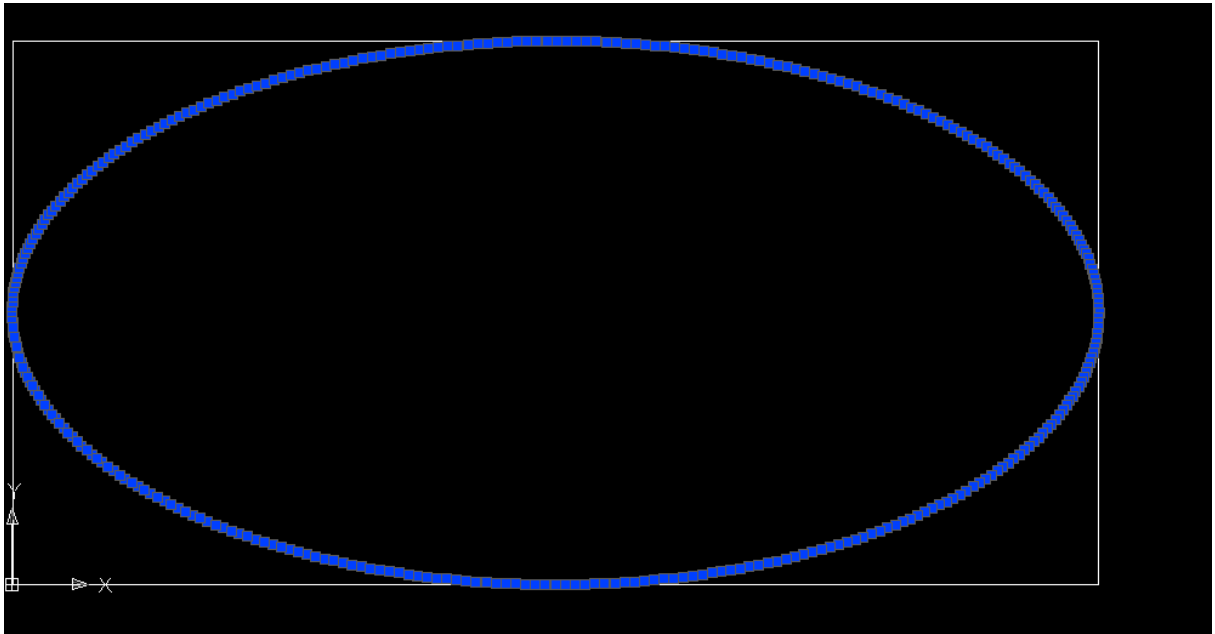


If you were to then change the NCVAR *NOPELLIPSE* to T in Router-CIM, erase your geoshape and then re-geoshape the geometry again, you would see this:



This should help straighten out some of the issues of cutting ellipses in Router-CIM. The reason some people prefer to use PELLIPSE in AutoCAD is because it makes a Polyline with ARCS instead of a lot of point to point data. With Polylines, you get much less code and the code is made up from arcs instead of a LOT of tiny line segments. However, the amount of the NC code is not really relevant if your geometry is not cut correctly and unfortunately there is no way to turn up the resolution of the Polyline created by AutoCAD when PELLIPSE is set to 1. You will get 16 arcs and that is it.

You may notice that if you touch the geoshape show above, with *NOPELLIPSE* in Router-CIM set to T, you will get many, many grips, like the drawing show below.



The sheer volume of the grips on the geoshape Polyline show that there is a great deal of resolution in this shape holding it closely to the true elliptical shape.

You can then cut this shape, and although it is more code, it will be exactly the shape you originally created.

Cutting a Spline in Router-CIM

Splines are special objects in that they have a continuously changing radius. Being such, they must be converted into arc and/or line segments to be able to produce NC code. If the AutoCAD Express Tools are installed, the FLATTEN command is available and Router-CIM will use it to convert the spline. You can also use the command manually in AutoCAD by typing FLATTEN at the command line or through the Express Tools menu or ribbon. If the FLATTEN command is not available, Router-CIM will convert the spline into small line segments for NC processing. If the FLATTEN command is failing to convert the spline, you may want to disable it from being used in Router-CIM Automation Suite.

To do so, add the variable *NOFLATNSPLINE* to the System variables with a value of T.

Please refer to the ['NCVar Editor'](#) section for information on how to add an NCVAR.

Sequence and Make Code

Router-CIM Toolbar: 

Router-CIM Ribbon: 

Keyboard: **SEQCODE**

Upon selecting Sequence from either the Control Panel or off of the Router-CIM toolbar, an NC Sequence Builder dialog window will appear. With this you have access to such features as Sequencing, Sorting, Reporting, Tool Life Study, Automatic Sub-Programming, NC Code Documentation, and Tool Path Editing all from this one screen

NC SEQUENCE BUILDER

Sequence NCP_SEQ_1 Cut Block Sequence **Clear All**

Sequence Definition

☒ Develop a Cut Sequence

☐ Sort Cuts by: Seq_Sort - Tool Sort, Closest Point **Select**

Tooling Requirements

☐ Make SubRoutines

☒ Develop Tooling Motions **Start and End Codes**

☐ Block Sequence ☒ Yes ☐ No

Information Development

☒ Make NC Code: ROUTER **Options...**

☐ Perform Machine Calc: Cut Time and Tool Changes **Select**

☐ Report Information: Router Report **Select**

☐ Edit Tooling Motions: Configuration Defined **Config...**

OK **Cancel** **Help**

Sequence Definition

Sequence NCP_SEQ_1 Cut Block Sequence **Clear All**

Sequence Definition

☒ Develop a Cut Sequence

☐ Sort Cuts by: Seq_Sort - Tool Sort, Closest Point **Select**

The NC Sequence Definition section allows you to develop a Cut Sequence and also to sort the cuts in the sequence according to several sorting keys available. There is also an option to clear out the sequence options so that you can start over or remove existing choices.

Clear All

Clear All

When selected after making a Sequence, this button will clear all fields. This is useful for having Sequence write only Report Information or Make NC Code again (instead of re-writing everything) or for just clearing the screen to give you a fresh start.

NC SEQUENCE BUILDER

Sequence NCP_SEQ_1 Cut Block Sequence [Clear All](#)

Sequence Definition

☒ Develop a Cut Sequence

☐ Sort Cuts by: Seq_Sort - Tool Sort, Closest Point [Select](#)

Tooling Requirements

☐ Make SubRoutines

☒ Develop Tooling Motions Start and End Codes

☐ Block Sequence ☒ Yes ☐ No

Information Development

☒ Make NC Code: ROUTER [Options...](#)

☐ Perform Machine Calc: Cut Time and Tool Changes [Select](#)

☐ Report Information: Router Report [Select](#)

☐ Edit Tooling Motions: Configuration Defined [Config...](#)

[OK](#) [Cancel](#) [Help](#)

Before Clear All is used.

NC SEQUENCE BUILDER

Sequence NCP_SEQ_1 Cut Block Sequence **Clear All**

Sequence Definition

☐ Develop a Cut Sequence

☐ Sort Cuts by: Seq_Sort - Tool Sort, Closest Point **Select**

Tooling Requirements

☐ Make SubRoutines

☐ Develop Tooling Motions **Start and End Codes**

☐ Yes ☐ No

☐ Block Sequence

Information Development

☐ Make NC Code: ROUTER **Options...**

☐ Perform Machine Calc: Cut Time and Tool Changes **Select**

☐ Report Information: Router Report **Select**

☐ Edit Tooling Motions: Configuration Defined **Config...**

OK Cancel Help

After Clear All is used.

Develop a Cut Sequence

NC SEQUENCE BUILDER

Sequence NCP_SEQ_1 Cut Block Sequence **Clear All**

Sequence Definition

☒ **Develop a Cut Sequence**

☐ Sort Cuts by: Seq_Sort - Tool Sort, Closest Point **Select**

Develops the Router-CIM database for selected cuts.

When you use 'Develop a Cut Sequence', you are defining all the cut cycle blocks, process list blocks and sequence blocks that will represent the basis of a sequence.

If you have selected an existing sequence that has developed tooling motions in it, the tooling motions are removed when the 'Develop a Cut Sequence' command is used.

An internal database is established when cut cycles are defined for use by Router-CIM. The database is referred to as a sequence knowledge. A sequence knowledge is an internal list that contains information about the selected cuts and any other Cut or Sequence commands that have attached additional data to the cuts. This database remains with the sequence in the form of an AutoCAD block in the current

drawing. When Router-CIM commands perform activity on a sequence, the current sequence database is used and updated.

If this is the only command chosen in the command options, the result would be a group containing the selected cut blocks.

If other command options are selected that require a new definition of cut cycles, the definition phase for the cuts will occur. This is true when you develop tool motions, sort the cuts, or define subroutines. The option dialog interface will disable these commands until you indicate that definition of cuts is to occur.

By default this option is checked and should normally be left that way.

Sort Cuts By



Redefines the order of a sequence definition

Sort Cuts is the re-arrangement of the cut blocks in the sequence based on a set of sorting rules. The sorting rules are defined in a sort task. The option to select the sort task is provided in the dialog interface.

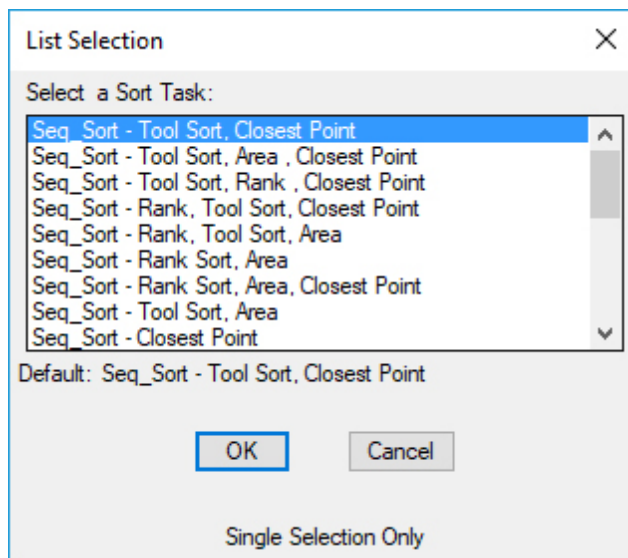
Since sorting can be used numerous times to determine the best sort, temporary vectors are generated between the cuts so that you can see the results. Do not confuse these temporary vectors with the index lines developed during tooling motions. These temporary lines will disappear upon the next redraw command.

If you are sorting and developing tooling motions, the temporary vectors are not generated since the index lines are produced during tooling motions development.

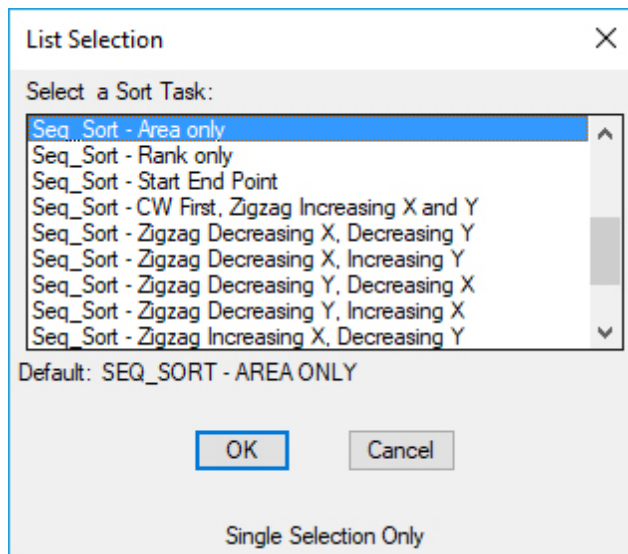
Sort simply rearranges the cut blocks in a sequence.

When you Sequence a Cut,

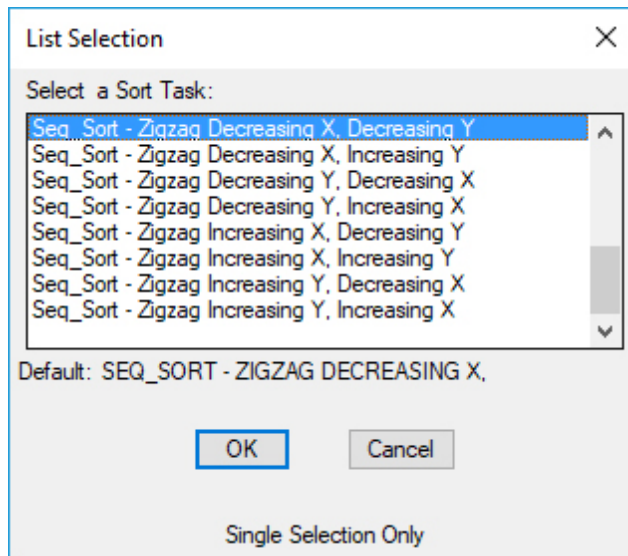
- 1) Pick the Sort Cuts by check box and click on the Select button.
- 2) Choose one of the Sort by Rank options as shown.



Sequence Sort Options-1



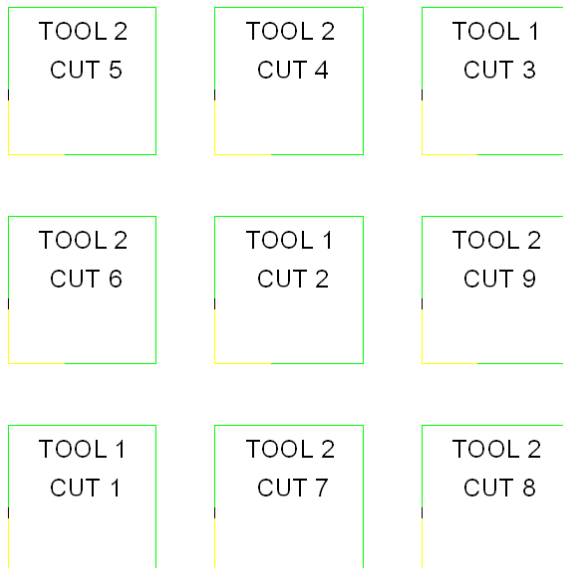
Sequence Sort Options-2



Sequence Sort Options-3

Seq_Sort -- Tool Sort, Closest Point

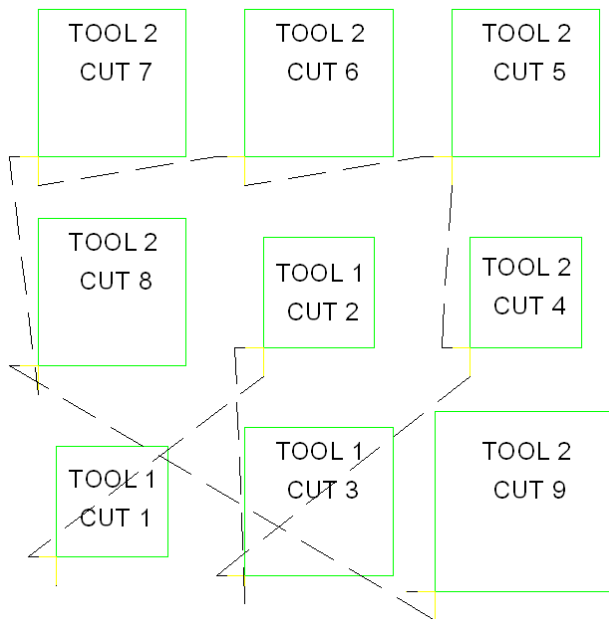
This will sort using the lowest number tool 1st, as many times as it can, sorting those cuts by closest point, then change tools to the next highest tool number and continue sorting by closest point, etc....



Tool Sort, Closest Point

Seq_Sort -- Tool Sort, Area, Closest Point

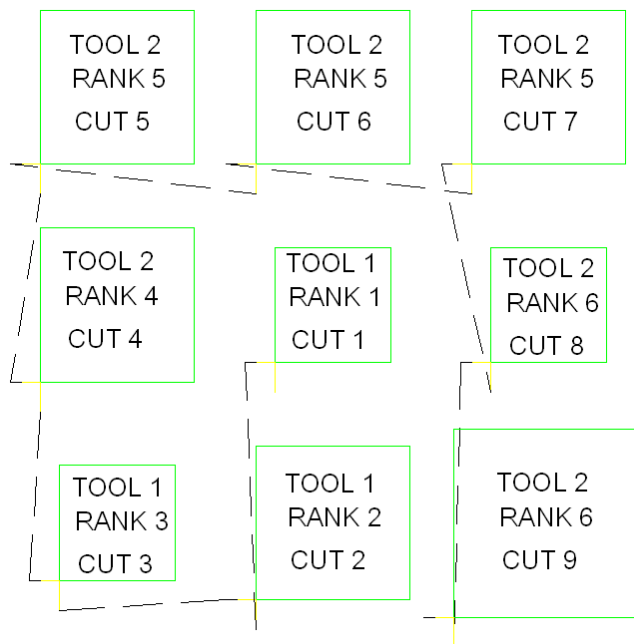
This will sort using tool number 1st, then area (smallest to largest), and then closest point.



Tool Sort, Area, Closest Point

Seq_Sort -- Tool Sort, Rank, Closest Point

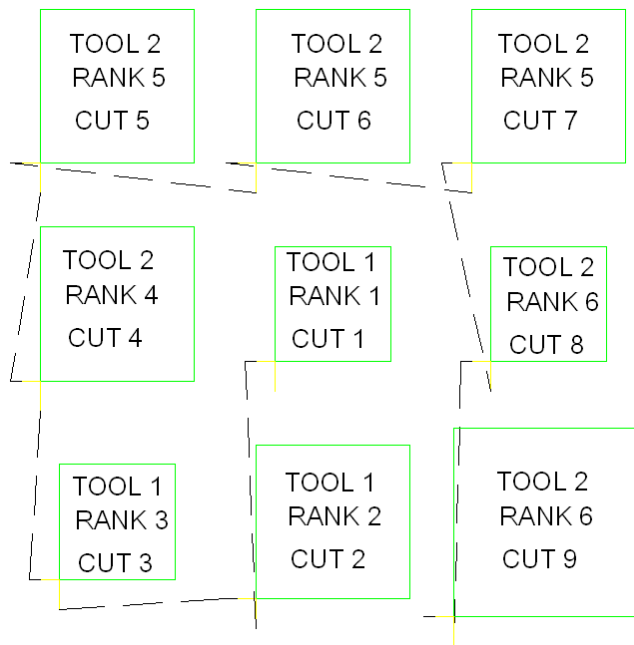
This will sort using lowest tool number 1st, then rank number, and then closest point. Looking at the example below, the cuts for tool 1 are gathered, then sorted by rank and placed in order according to closest point. Next, the tool 2 cuts are gathered, then sorted by rank and then selected according to closest point as some of them are the same tool and rank.



Tool Sort, Rank, Closest Point

Seq_Sort -- Rank, Tool Sort, Closest Point

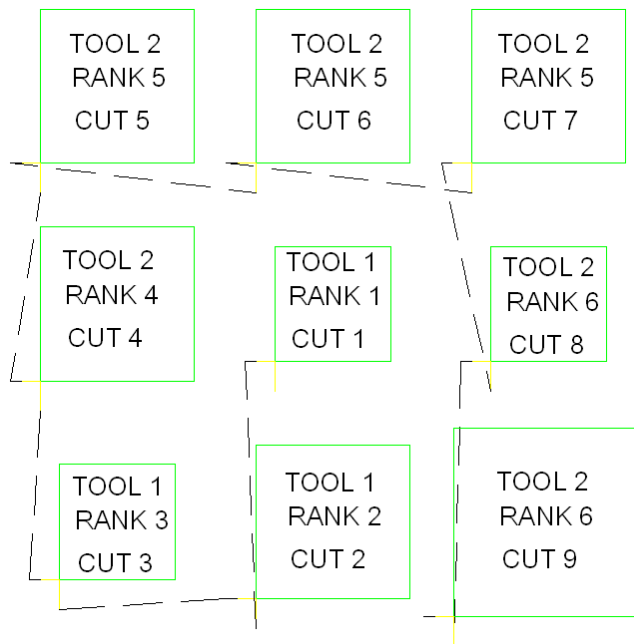
This will sort using rank 1st, then lowest tool number, and then closest point. In fact the particular example shown below the cuts will be placed in the same order as the sort above. All cuts are sorted by rank first, and then within the ranked cuts, the tool numbers are sorted, lowest first. Last the cuts are sorted according to location from one cut to another within the rank and tool sort.



Rank, Tool Sort, Closest Point

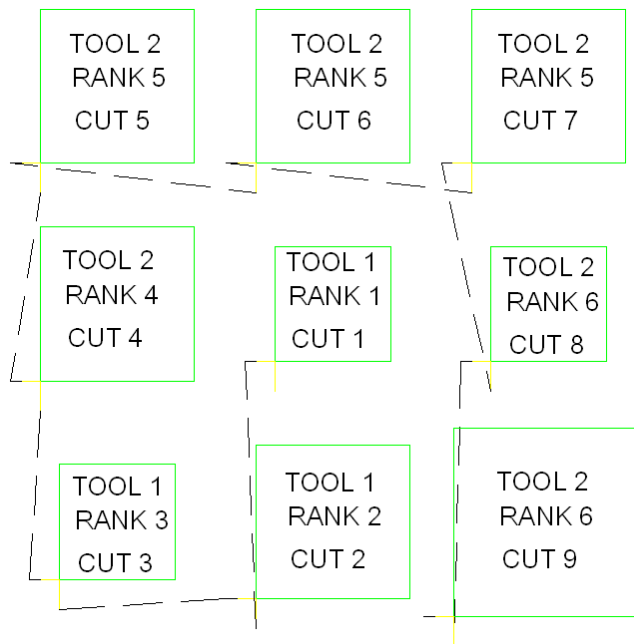
Seq_Sort -- Rank, Tool Sort, Area

This will sort using rank 1st, then lowest tool number, then area (smallest to largest). Once again, setting the Sequence up in the same order as the last two since the area sort is last, cut 1, cut 2, and cut 3 are placed in Rank and then Tool order first and since there is only 1 of each rank, the area sort is really canceled out. The tool 2 cuts (4-9) are sorted by rank and since there is only one cut with rank 4, it has to be first. There are 3 cuts with rank 5, but they are all the same tool number and area, so the sequence places them in an order (any order for these cuts would be valid, even if it was not according to closest point...we just got lucky there) and finally there are two rank 6 cuts and both are made with the same tool number, but one is smaller than the other so it must come first.

**Rank, Tool Sort, Area**

Seq_Sort -- Rank Sort, Area

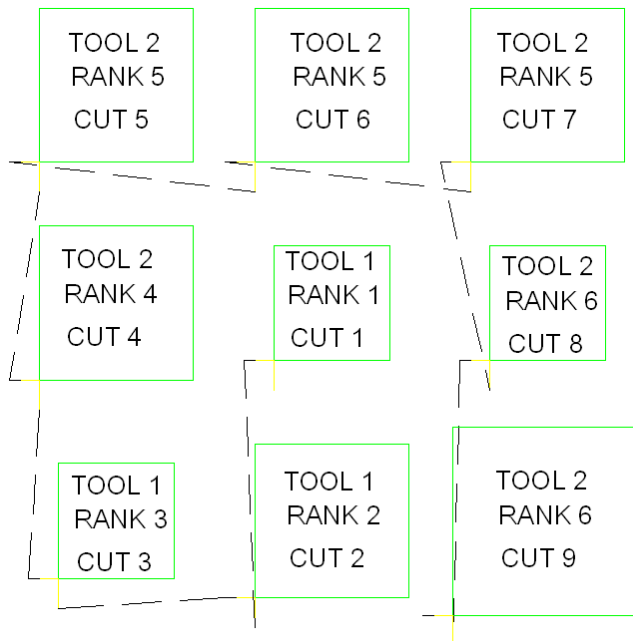
This will sort using rank 1st, then area (smallest to largest). This order is still similar to the last few, as the rank sorting is the first key. Cuts 1-4 are ordered according to rank, then cuts 5-7 are made in any order since they are the same rank, and area. Finally cuts 8 and 9 are made in area order, since they are the same rank.

**Rank Sort, Area**

Seq_Sort -- Rank Sort, Area, Closest Point

This will sort using rank 1st, then area (smallest to largest), then closest point. This is once again the same order as the last few (you couldn't get this order in a real job, honest). Since the cuts 1-4 are in rank order and there are no duplicates, we move on to cuts 5-7 which are the same rank, and same area.

They get sorted according to closest point (remember before we just got lucky, these cuts only have to appear in this order during this sort key). Finally, cuts 8 and 9 are made with the same rank, so they get sorted according to area and the sequence is finished.

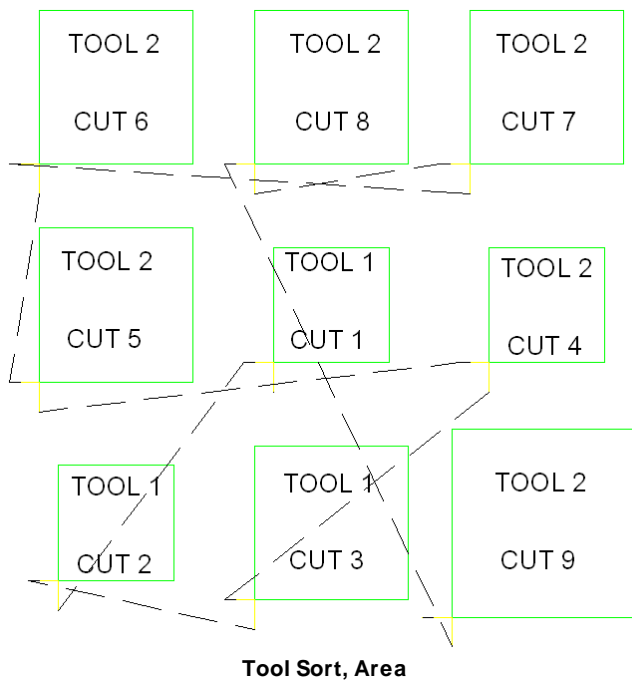


Rank Sort, Area, Closest Point

Seq_Sort -- Tool Sort, Area

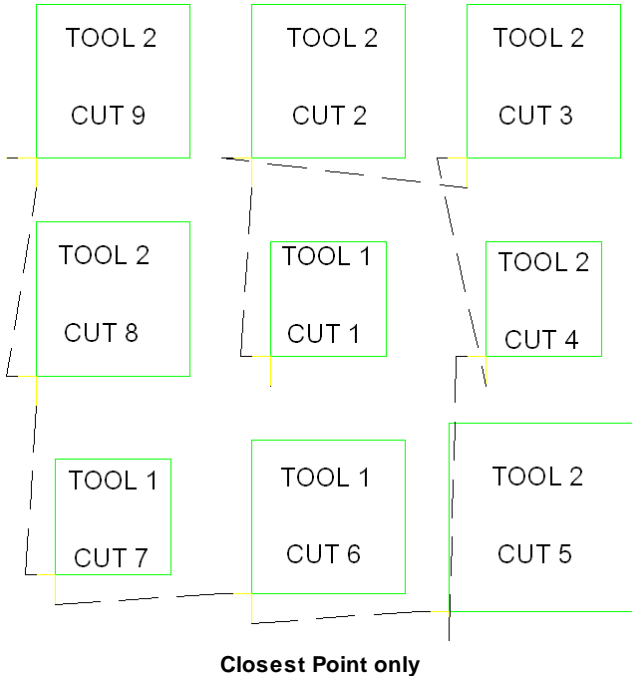
This will sort using lowest tool number 1st, then area. Since the location of the cuts is not a factor, they are tool sorted, then sorted according to size. Cuts 1-2 are the same tool and area, cut 3 is made with tool 1 and is larger than cuts 1 and 2 so it comes after. Cut 4 is the smallest tool 2 cut, so it is next.

Cuts 5-8 are all the same tool and area, so they get picked in no particular order, then on to cut 9 which is the largest cut made with tool 2 so it comes last.



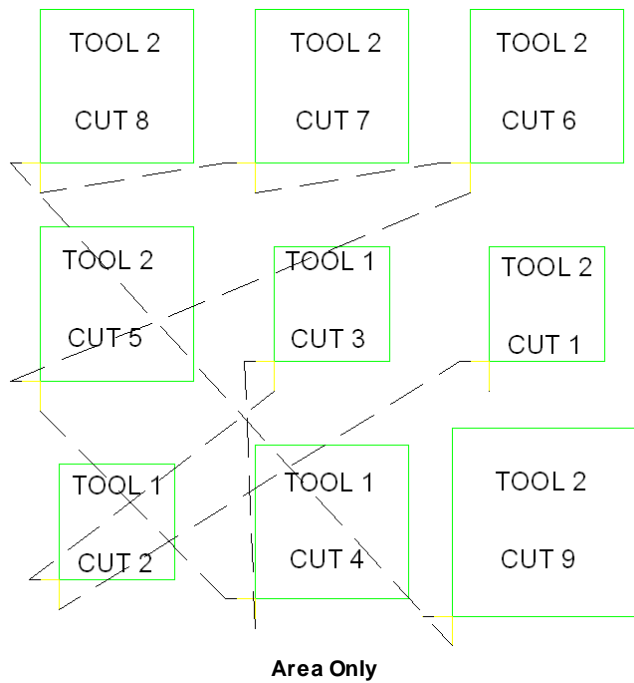
Seq_Sort -- Closest Point

This will sort using closest point only. The tool number, area, rank, etc are all ignored.



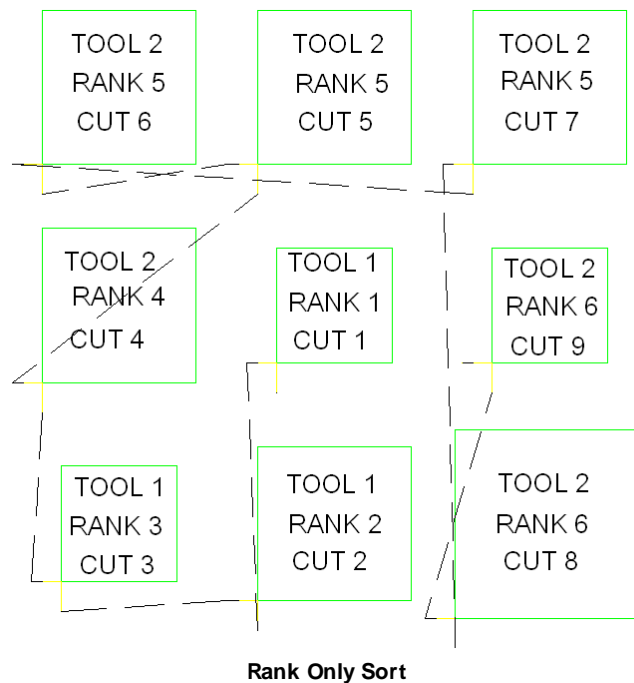
Seq_Sort -- Area Only

This will sort using area only. It will start with the smallest and work towards the largest shape. No other factors are considered.



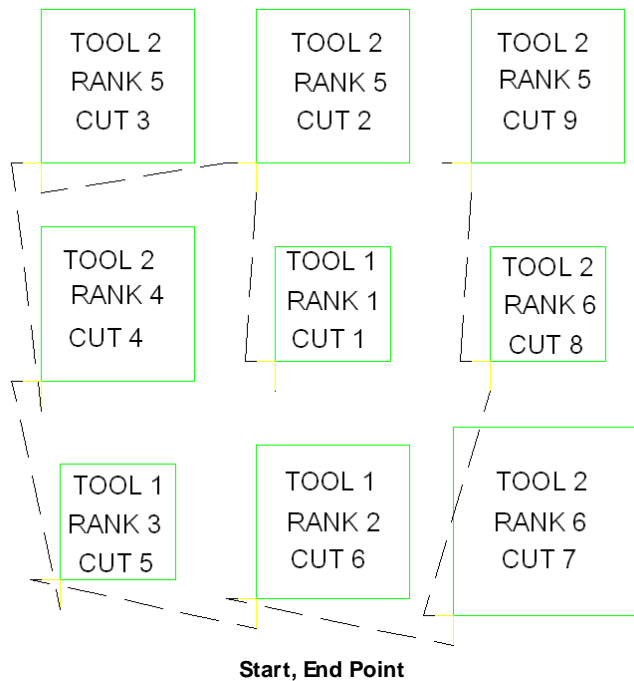
Seq_Sort -- Rank Only

This will sort using rank only. The lowest rank is first, progressing toward the highest. Decimal and Integer numbers are allowed. There are no other considerations.



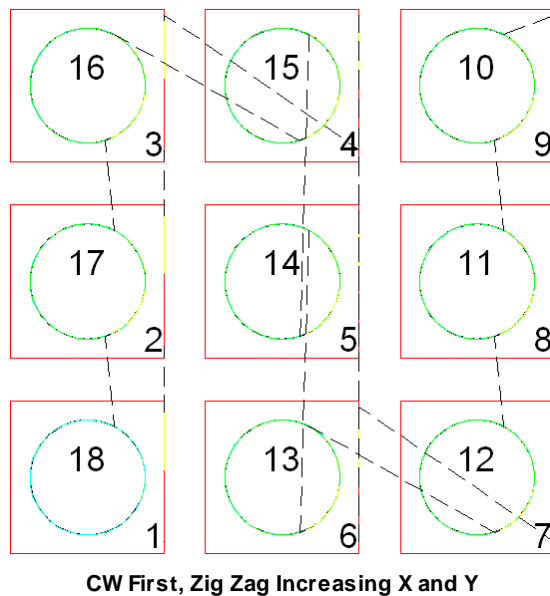
Seq_Sort -- Start End Point

This option will sort by moving from the end point (lead-out) of a cut to the next closest start point (lead-in).



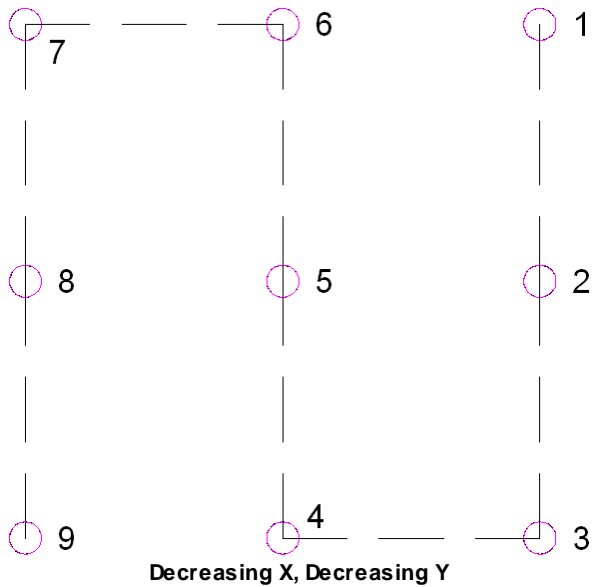
Seq_Sort -- CW First, Zig Zag Increasing X and Y

This will sort cuts made in a clockwise direction starting with the lowest X and Y and cut in Increasing X and Y positions until it gets to the end of the CW tool paths and then reverse its direction walking the Sequence back to where it started.



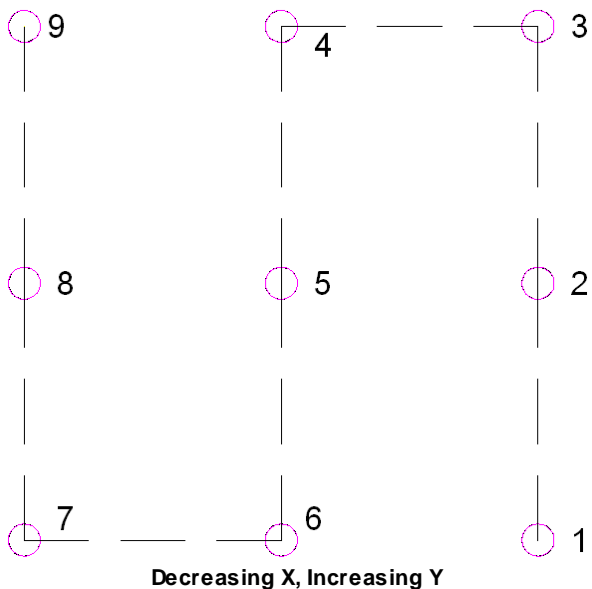
Seq_Sort -- Zigzag Decreasing X, Decreasing Y

This will sort cuts starting at the farthest lead-in in X and Y and then proceed to the next closest lead-in while decreasing in X and decreasing in Y values.



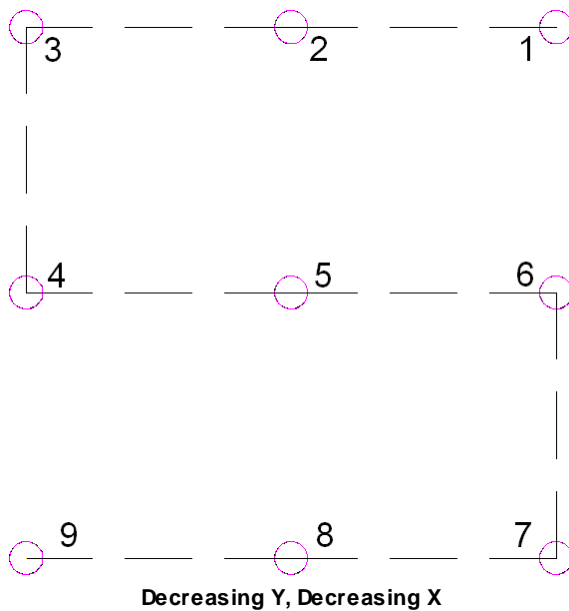
Seq_Sort -- Zigzag Decreasing X, Increasing Y

This will sort cuts starting at the farthest lead-in in X, but lowest Y and then proceed to the next closest lead-in while decreasing in X and increasing Y values.



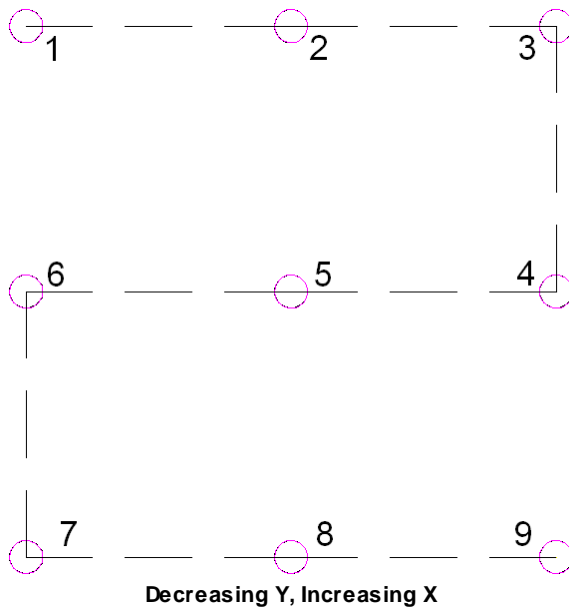
Seq_Sort -- Zigzag Decreasing Y, Decreasing X

This will sort cuts starting at the farthest lead-in in X and Y and then proceed to the next closest lead-in while decreasing in Y and decreasing in X.



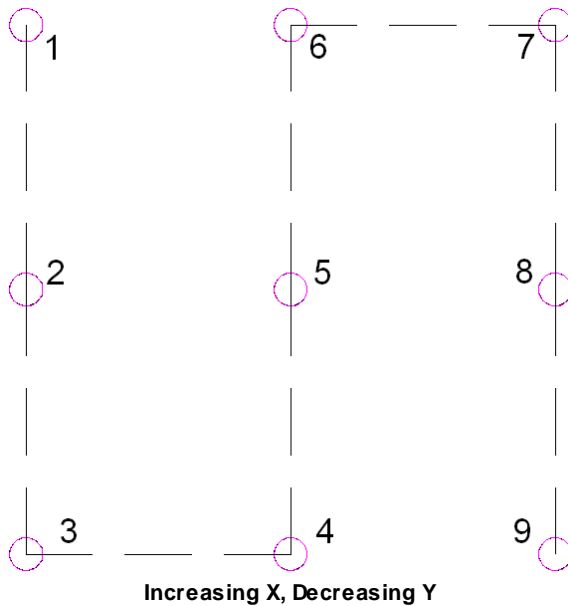
Seq_Sort -- Zigzag Decreasing Y, Increasing X

This will sort cuts starting at the farthest lead-in in Y, but closest in X, and then proceed to the next closest lead-in while decreasing in Y and increasing in X.



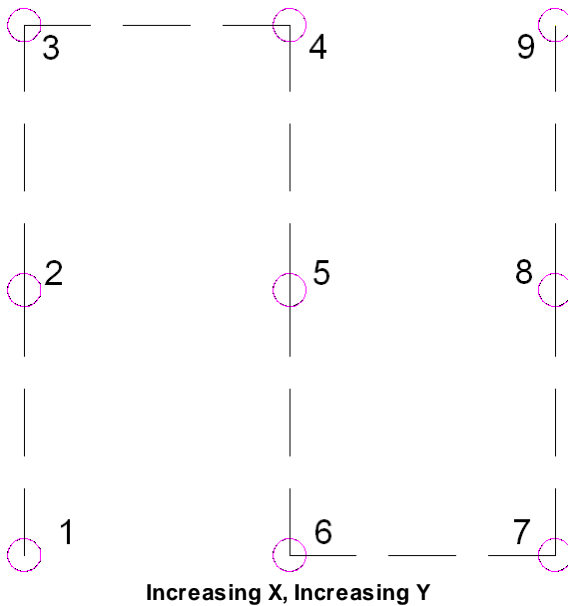
Seq_Sort -- Zigzag Increasing X, Decreasing Y

This will sort cuts starting at the closest X and farthest Y lead-in, then proceed to the next closest lead-in while increasing in X and decreasing in Y.



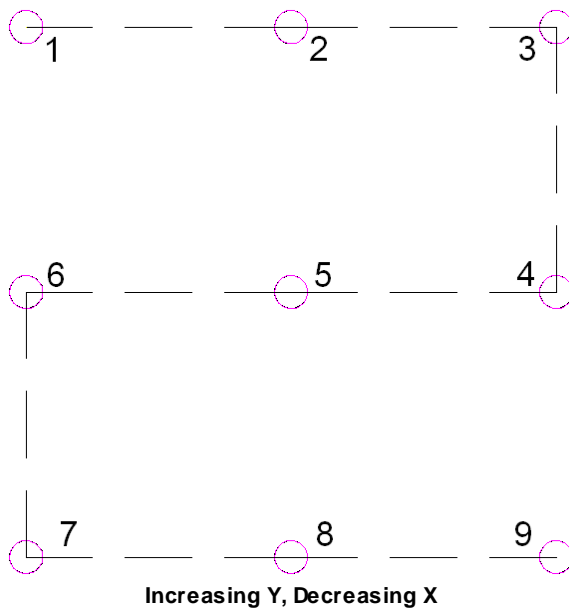
Seq_Sort -- Zigzag Increasing X, Increasing Y

This will sort cuts starting at the closest lead-in in both X and Y and then proceed to the next closest lead-in while increasing in X and Y values.



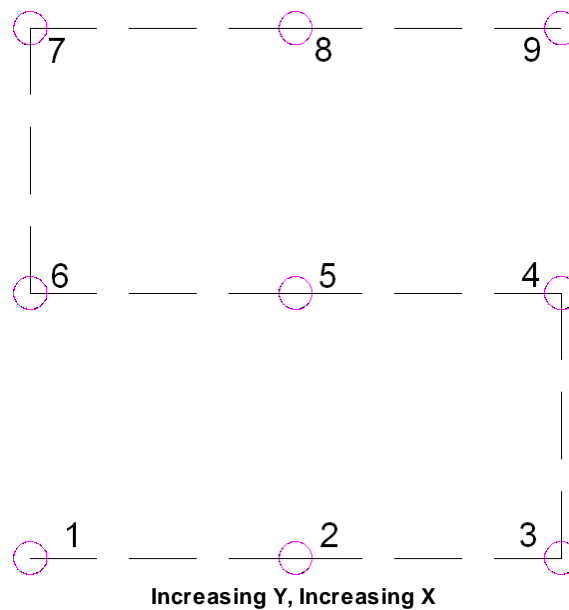
Seq_Sort -- Zigzag Increasing Y, Decreasing X

This will sort cuts starting at the closest X, but furthest Y lead-in and then proceed to the next closest lead-in while increasing in Y and decreasing in X.

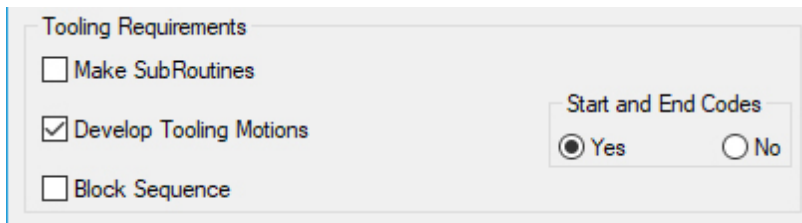


Seq_Sort -- Zigzag Increasing Y, Increasing X

This will sort cuts starting at the closest lead-in in both X and Y and then proceed to the next closest lead-in while increasing in Y and increasing X.



Tooling Requirements



Tooling Requirements

☐ Make SubRoutines

☒ Develop Tooling Motions

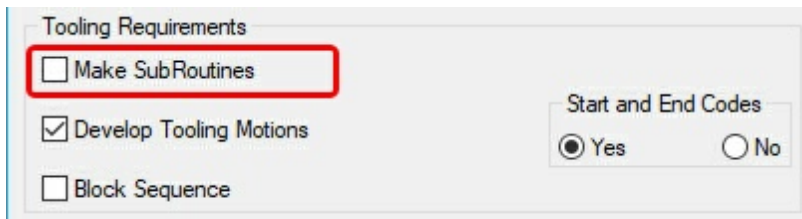
☐ Block Sequence

Start and End Codes

☒ Yes ☐ No

The Tooling Requirements section will cover items that affect the elements within the tool paths, like whether or not to use Sub Programs or Starting and Ending safety blocks. Each section will be discussed in more detail.

Make Subroutines



Tooling Requirements

☐ Make SubRoutines

☒ Develop Tooling Motions

☐ Block Sequence

Start and End Codes

☒ Yes ☐ No

Make subroutines is the development of subroutine information that is attached to the sequence and to each of the cut blocks. This information is then used by the task functions in 'Develop Tooling Motions' to form the correct commands representing subroutine construction.

The developed subroutine information is not easily seen since it resides in the Router-CIM internal database (NCPS) as lists of information to be used later by other commands. A user will not see the result until tooling motions are developed and NC code generated. A programmer can look at the results by examining the supplied NCPS Variable Definitions during the develop tooling motions command.

Variables

There is an NCVAR (under System) called Auto_Sub_Num. When this variable is set to T for True (on), SubRoutines will increment in hundreds, starting at 100 and incrementing by 100 for each sub. If this variable is changed to nil for false (off), you will be prompted at the command line for the Sequence ID number (put in any number as you would normally to call out SubRoutines).

Sub-Programming

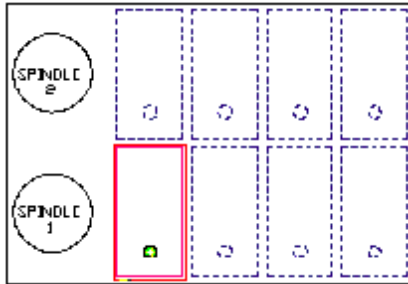
Sub-Programming is a method used to aid in the editing of NC Code and to minimize redundant code.

To generate a program using Incremental Sub-Programming, Copy an existing Cut to the locations where you want to call up the Sub-Program. Each copied Cut will be part of the Sub-Program.

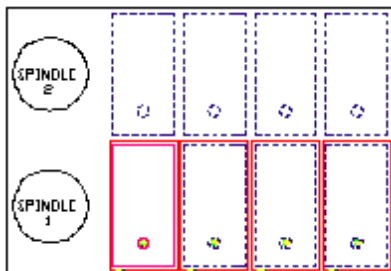
NOTE: The Make SubRoutines box must be checked in the NC Sequence Builder (sequence command), for the program to generate Sub-Programs. If the box is left un-checked, then no Sub-Programs will be generated, even if the proper sub routine is followed. The code produced will repeat for each occurrence of the Cut.

EXAMPLE: Make 8 parts using spindles 1 and 2 using ½" end mill in each spindle.

STEP 1. Geoshape and Cut parts. Do not include any slaving spindle parts or parts using the same spindle(s) that the Cut can be copied to. The dashed lines(parts) are not included in the pick because they are either slaving spindle parts (parts in line with Spindle 2) or the cuts can be copied from the first part because they use the same spindle and they have the same part profile.



STEP 2. Copy or Array Cut(s) to new locations. In this example only the Cuts that spindle 1 will be cutting are copied. (Remember, Spindle 2 is slaving whatever Spindle 1 is doing).

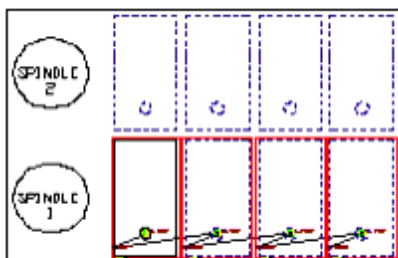


A Sub-Program is required for each different spindle (or spindle combination) used. There are an almost endless variety of Cut combinations that you can use to make multiple parts.

STEP 3. Sequence the parts. Select all parts in order to be Cut, or if you are going to select Sort Cuts By on the NC Sequence Builder dialog window, select the parts by putting a window around all the parts (this includes copied CUTS).

STEP 4. Press <Enter>. If selecting manually, this step is selection sensitive: the order in which you select the Cuts will be the order in which they will be cut in the program.

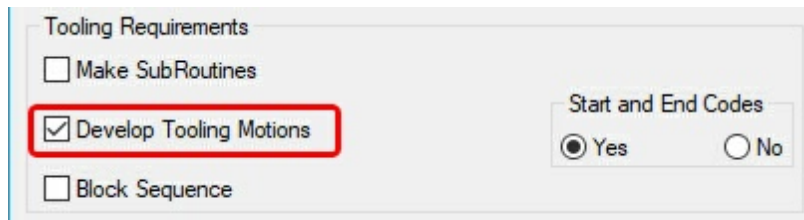
STEP 5. The NC Sequence Builder dialog window will appear. Select and enter all appropriate information required. Make sure to select the Make Subroutines box. When complete, click on <OK>. Now your drawing should be back on the screen and the command prompt will be asking you to select single cuts for sub-program creation. Always press <Enter> to this prompt and the Sequence command will begin gathering all information you entered previously in the NC Sequence Builder and will create NC code.



The code will then appear on the screen for you to view. (Only if you selected Yes to the Preview Code option under the NC Codes Options dialog window).

NOTE: After entering in all the information required in the NC Sequence Builder, select <OK>. If you are making Incremental SubRoutines, you will just press <Enter> at the next prompt that follows, if you are making individual SubRoutines for a given part(s), you will select that part(s) individually now and then press <Enter>.

Develop Tooling Motions



Develops the tooling motions between cut cycles such as Index moves.

Develop Tooling Motions is the adding of NC objects into the sequence that represent tooling motions and commands required to traverse from one cut cycle to another. Typically these objects include index motions and tool change commands.

The option for the use of start and end codes is supplied to control what codes the task functions supply at the start and end of the sequence definition.

This process is a loop over all the cut blocks defined to the sequence. Two functions are performed. The SEQTSK1 task function develops NC objects that appear at the beginning of each cut block. The SEQTSK2 task function develops NC objects that appear at the end of each cut block.

If SEQTSK1 and SEQTSK2 functions are not defined, the Process MAKE Simulator is used to loop over all the cut blocks.

All the activity of determining what is appropriate NC objects is made through the use of task functions. These functions used by SEQTSK1 and SEQTSK2.

See NCPS Variable Definitions for programming variables.

The result of 'Develop Tooling Motions' is an expanded sequence with more objects in the appropriate locations in the sequence. Usually this can be seen by the index lines developed between each of the cut blocks.

Block Sequence

Tooling Requirements

☐ Make SubRoutines

☒ Develop Tooling Motions

☐ Block Sequence

Start and End Codes

☒ Yes ☐ No

Block Sequence converts a sequence group into a NC object block

Block Sequence develops a block using the sequence information.

This is normally done when a sequence is fully developed with tooling motions and no start and end codes. The resulting blocked sequence is to be used as a cut block for another sequence. This is known as a nested sequence.

Sequences do not have to be blocked to produce NC code.

Start and End Codes

Tooling Requirements

☐ Make SubRoutines

☒ Develop Tooling Motions

☐ Block Sequence

Start and End Codes

☒ Yes ☐ No

Yes is the default.

Yes will give you the safety blocks in the beginning and at the end of the program. It will also provide the tool calls for each cut. Setting this feature to No will not provide any safety blocks or tool changes, but just the XYZ motions of the cut.

For example, all of the cycles to be cut by the same spindles at the same time can be grouped into a Sequence without Start and End Codes (by selecting No). Each group of Sequences can then be grouped again into a final Sequence with Start and End codes.

Here is the code for a sequence with Start and End Codes set to NO:

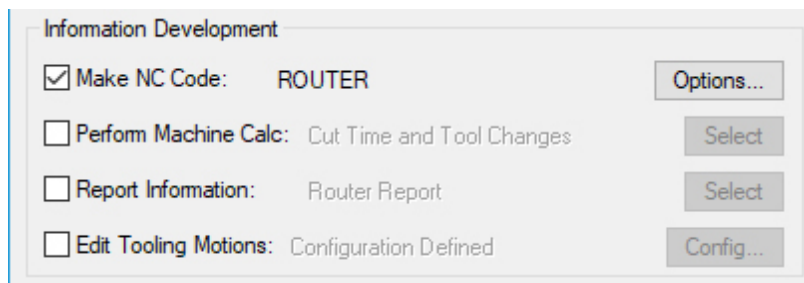
```
%
:1
N1G41D01G01Z-.75F200.
N2G03X3.Y-.249I-.5F500.
N3G01X.5
N4G02X-.249Y.5J.749
N5G01Y3.5
N6G02X.5Y4.249I.749
N7G01X5.5
N8G02X6.249Y3.5J-.749
```

```
N9G01Y.5
N10G02X5.5Y-.249I-.749
N11G01X3.
N12X2.5F500.
N13G03X2.Y-.749J-.5
N14S18000
N15G00G43H0Z.25
N16G40G00Y-1.024
%
```

Here is the same tool path with the Start and End Codes set to YES. The code in red is what Start and End Code added to the program.

```
%
:1 (START AND END CODES ON)
N1G00G17G20G28G40G80G91Z0M5
N2G90
N3G52X0Y0Z0
N4G08P1
N5M08
N6(ROUTER-BIT .5 DIA.)
N7G28G91Z0M05
N8G90T2001M06
N9T102
N10M03S18000
N11G00G17G55X3.5Y-1.024
N12G00G43H1Z.25
N13G41D01G01Y-.749F200.
N14Z-.75
N15G03X3.Y-.249I-.5F500.
N16G01X.5
N17G02X-.249Y.5J.749
N18G01Y3.5
N19G02X.5Y4.249I.749
N20G01X5.5
N21G02X6.249Y3.5J-.749
N22G01Y.5
N23G02X5.5Y-.249I-.749
N24G01X3.
N25X2.5F500.
N26G03X2.Y-.749J-.5
N27G00Z.25
N28G40G00Y-1.024
N29G28G91Z0M5
N30G28G91X0M09
N31G90
N32G52X0Y0Z0
N33G08P0
N34M30
%
```

Information Development



The screenshot shows a dialog box titled "Information Development". It contains four rows of settings, each with a checkbox, a text field, and a button:

Option	Value	Button
<input checked="" type="checkbox"/> Make NC Code:	ROUTER	Options...
<input type="checkbox"/> Perform Machine Calc:	Cut Time and Tool Changes	Select
<input type="checkbox"/> Report Information:	Router Report	Select
<input type="checkbox"/> Edit Tooling Motions:	Configuration Defined	Config...

The Information Development section of the Sequencer allows for the creation of the NC Code file, reporting, and tool emulation and editing.

Make NC Code

Make NC Code develops an NC Code file for a selected sequence

To define the options for developing the NC Code see the section NC Code Options section.

This command uses the programmable postprocessor through the task function `ncpp_seq_nccode`.

When selected, click on the **Options** button to the right and a NC Code Options dialog window will appear. From this screen you will have the following options:

NC Code Options

Postprocessor Name: ROUTER Browse

Output File Name: drawing1.out Browse

Job Identification Number:

Material X Dimension:

Material Y Dimension:

Material Thickness:

MHS Bunk Number:

☐ Manual origin ☒ Use line numbers

Starting line number:

Line number increment:

Generate MCU Comments

☐ Yes ☒ No

Preview NC Code

☒ Yes ☐ No

Program Comments: Edit

NC Code Generation Options

☐ Acceleration/Deceleration Configure...

OK Cancel

Most of these settings have NC System Variables that control their use.

- *POST* - postprocessor name
- *NCFIL* - output filename
- *NCFILEXT* - output filename extension
- *NCFILLOC* - output file directory
- *JOBID* - job identification number
- *VIEWCODE* - Preview code option
- _ACC_DEC - Acceleration/Deceleration option

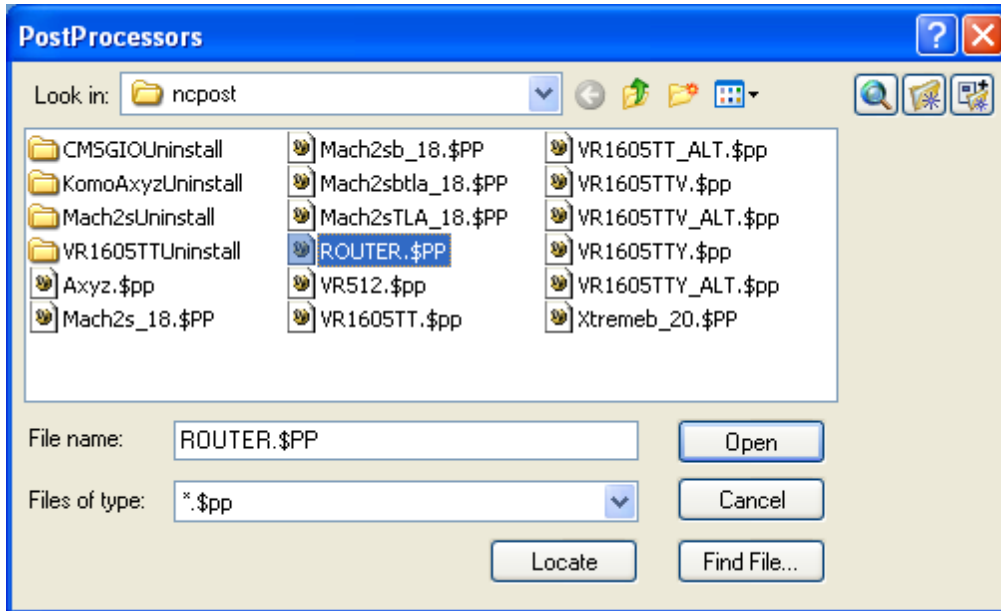
Any or all of these can be defaulted by using the NCVAR command.

Make NC Code Options

Make NC Code Options provide all the machine specific options available during the creation of the sequence.

Postprocessor Name

A default name follows this field. If you want to change the Postprocessor, click on the **'Browse'** button to the right and you can select a different Postprocessor. The software is configured to direct you to the Ncpost folder if you do click on the browse button.



Output File Name

The default shows you the Path where the code will be written to and the name of the NC Code file (it defaults to the name of your current drawing) with an .OUT extension. You can change the name of the file or the extension to anything you want (as long as the Path is valid.)

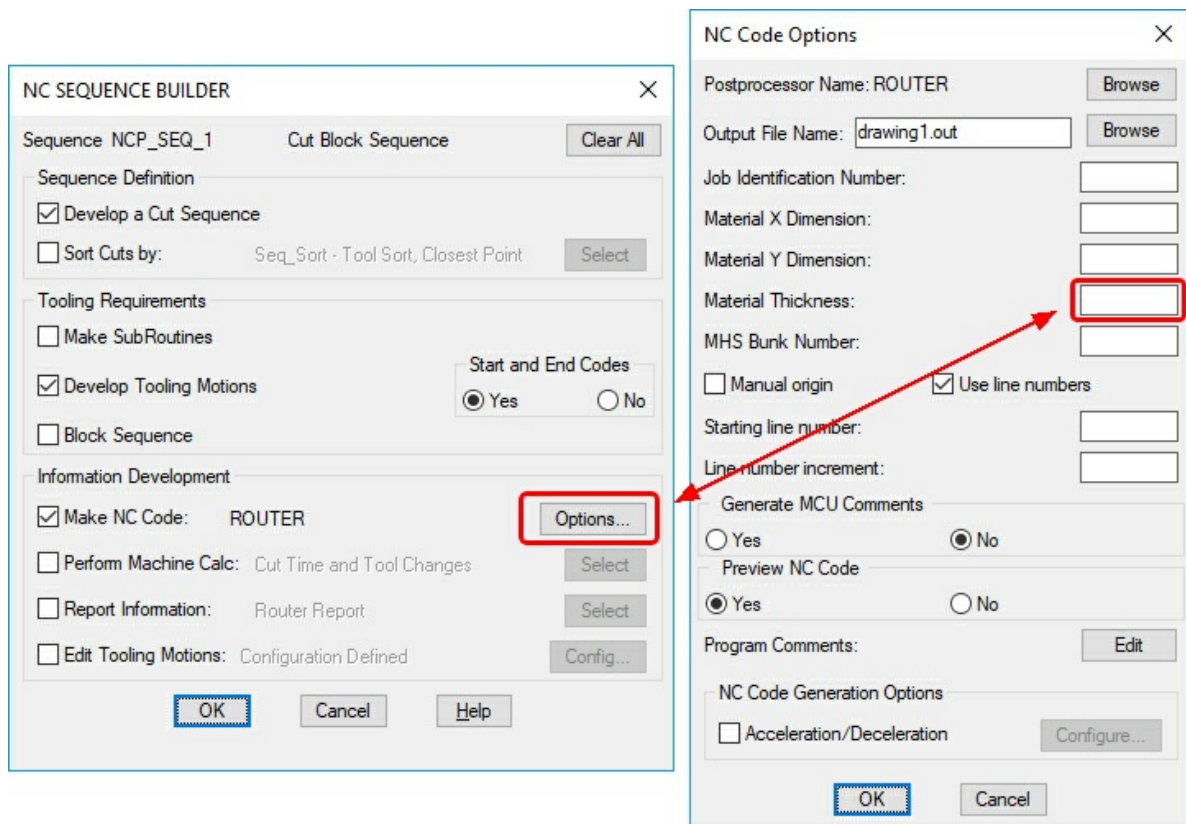
Job Identification Number

This field is where you will enter the 4-digit Main Program Number (or whatever numbering system your machine typically uses).

Material Thickness Comp.

If you program Z0 off the top of the spoil board instead of using the top of your part, you still program as if the top of part is Z0. To set Router-CIM for spoil board as Z0, put in your material thickness in the Material Thickness Compensation field shown below. The interface below is arrived at after a cut is made and you are ready to Sequence.

From the Sequence screen you choose the Options button to the right of "Make NC Code."



If you program 0,0 as the top of your part, you can leave this field blank.

Manual origin

When selected, you can tell Router-CIM where you want 0,0 to be on the screen (it can be anywhere) instead of it being at the lower left corner (by default). This is useful if you are programming multiple parts on your screen, in that you won't have to move the various parts to 0,0 then move them back out of the way to move another part down to 0,0. If you turn Manual origin on, you will be prompted for where you want 0,0 to be. (You can change the 0,0 location as many times as you need).

Use line numbers

If this box is checked, the NC Code produced will start with line number N1 and increment or increase by 1 for each line.

If this box is not checked, The NC Code produced will have no line numbers. This option might be chosen if you wish to save memory space at the controller.

Starting line number

If you do not want your Starting line number to be 1, you may change it to what you wish in this first field (100 for example).

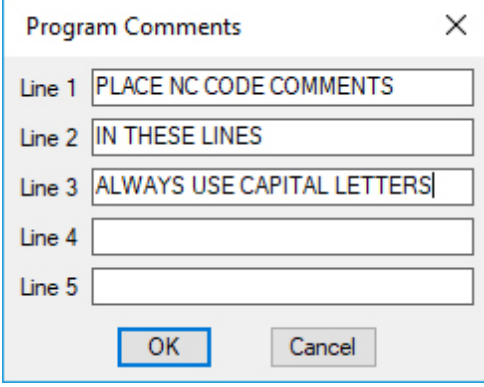
It will still increase or increment by 1.

Line number increment

If you want to change how the numbers increment or increase, you may do this in the Line number increment field. (You may want the numbers to increase by 10's rather than 1's).

Program Comment

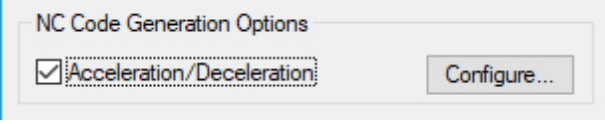
Click on the Edit button to the right and a dialog window will appear. You are given five lines where you can enter any comments you wish to appear in the program.

A dialog box titled "Program Comments" with a close button (X) in the top right corner. It contains five text input fields labeled "Line 1" through "Line 5". Line 1 contains the text "PLACE NC CODE COMMENTS", Line 2 contains "IN THESE LINES", and Line 3 contains "ALWAYS USE CAPITAL LETTERS". Lines 4 and 5 are empty. At the bottom of the dialog are two buttons: "OK" and "Cancel".

Preview Code

This field gives the options, Yes or No, of whether or not you want the code to pop up on the screen for viewing after it has been written.

Acceleration/Deceleration

A dialog box titled "NC Code Generation Options". It contains a checked checkbox labeled "Acceleration/Deceleration" and a button labeled "Configure..." to its right.

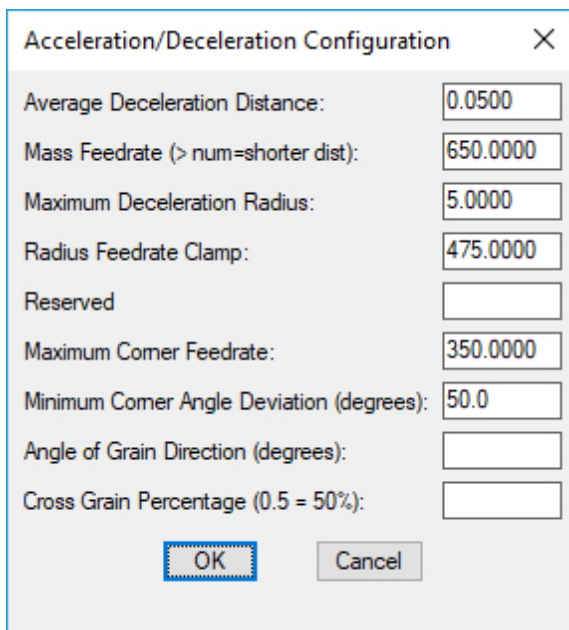
You can apply Acceleration and Deceleration to your NC program by selecting this option. Each motion in the NC Code will be examined for feed requirements and adjusted if necessary. The configuration button allows for the changes to the parameters that control acc/dec.

ACC 'n DEC anticipates change in direction, short moves and/or tight corners, and automatically inserts slow down or control moves into the machine code. ACC 'n DEC was designed to enhance the performance on all machine tools regardless of the controller's ability.

Select the option to turn on Acceleration/Deceleration, and then click on the Configure button to the right to display the Configuration options.

An Acceleration/Deceleration Configuration dialog window will appear.

A brief description for each field follows:



Parameter	Value
Average Deceleration Distance:	0.0500
Mass Feedrate (> num=shorter dist):	650.0000
Maximum Deceleration Radius:	5.0000
Radius Feedrate Clamp:	475.0000
Reserved	
Maximum Corner Feedrate:	350.0000
Minimum Corner Angle Deviation (degrees):	50.0
Angle of Grain Direction (degrees):	
Cross Grain Percentage (0.5 = 50%):	

Average Deceleration Distance

Distance to travel to allow for the Mass Feedrate to occur. This is an average because some rounding can occur.

The NCVAR is _ACC_DECDIST.

Mass Feedrate(> num=shorter dist)

The decrease in feedrate from one element to the next over the distance specified by the Average Deceleration Distance. A ratio defines the deceleration scale.

The NCVAR is _ACC_DECFEED.

Maximum Deceleration Radius

This parameter sets the maximum radius that can be cut at the Radius Feedrate Clamp. Any radius less than this will cause Acc/Dec to be applied.

The NCVAR is _ACC_CLAMPRADIUS.

Radius Feedrate Clamp

This is the maximum feed rate that is used on the largest arc setting set in Maximum Deceleration Radius.

The NCVAR is _ACC_CLAMPFEED.

Reserved

Various values are stored here when the system is in use. Not user configurable.

Maximum Corner Feedrate

This is the maximum feedrate that will cut a corner with an angular deviation greater than the Minimum Corner Angle Deviation so that the cutter will not over/under shoot the corner.

The NCVAR is _ACC_CLAMPMIN.

Minimum Corner Angle Deviation (degrees)

Minimum Angle Deviation of a corner when acc/dec is applied. Any angle deviation between elements or arc sweep that is greater than this value will cause acc/dec to be applied.

Router-CIM will automatically vary the feed rate on angles that are equal or smaller than the angle size set in this field.

The NCVAR is `_ACC_ANGDEV`.

Angle of Grain Direction (degrees)

This is the angle of the grain on the part in degrees. Angles from grain direction are cut at a feedrate scale based on the Cross Grain Percentage.

The NCVAR is `_ACC_GRAINDIR`.

Cross Grain Percentage (0.5=50%)

The percentage of current feedrate to cut 90° across the grain of the part. The value input here (must be in decimal form) will give you a feed rate equal to the percentage, entered in this field, of the maximum feed rate set in the Control Panel.

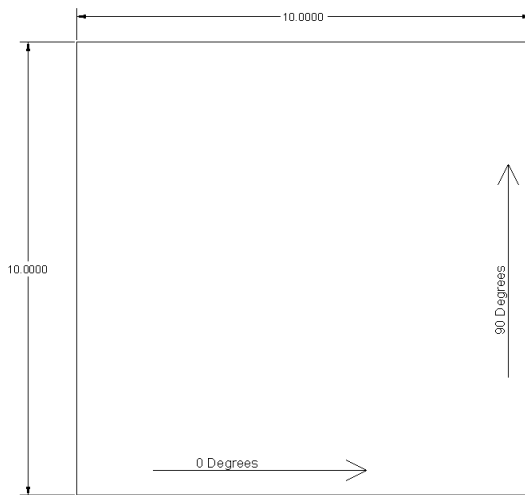
The NCVAR is `_ACC_CROSSGRAINPER`.

How Acc-n-Dec works

To show how Acc-n-Dec works, we will use a simple example.

EXAMPLE:

A part with an outside corner that has a 90° change between two elements that define the corner. Each element is a line 10 units long. First element is at 0 degrees and the second element is at 90 degrees.



Acc-n-Dec part

Current setting are:

```

_ACC_DECDIST = 1.0
_ACC_DECFEED = 100
_ACC_CLAMPIN = 5
_ACC_ANGDEV = 15
_ACC_GRAINDIR = 0.0
_ACC_CROSSGRAINPER = 1.0
_ACC_FEEDRATE = 200 (normal programmed feedrate)

```

Results:

The angle deviation is greater than `_ACC_ANGDEV` and the current feedrate exceeds the maximum corner feedrate expressed in `_ACC_CORNFEED`. Therefore, acc/dec will be applied.

The `_ACC_DECDIST/_ACC_DECFEED` ratio is applied to the current feedrate to determine a distance from the corner where the `_ACC_CORNFEED` will be applied. In this example, the distance from the corner is

$DIST = (\text{Programmed Feedrate} - \text{Max Corner Feedrate}) * \text{Average Deceleration Distance} / \text{Mass Feedrate}$ **OR**

$DIST = (/ (* _ACC_DECDIST (- _ACC_FEEDRATE _ACC_CLAMPMIN)) _ACC_DECFEED)$ **OR**

$DIST = (/ (* 1.0 (- 200.0 5.0)) 100.0) = 1.95 \text{ units}$

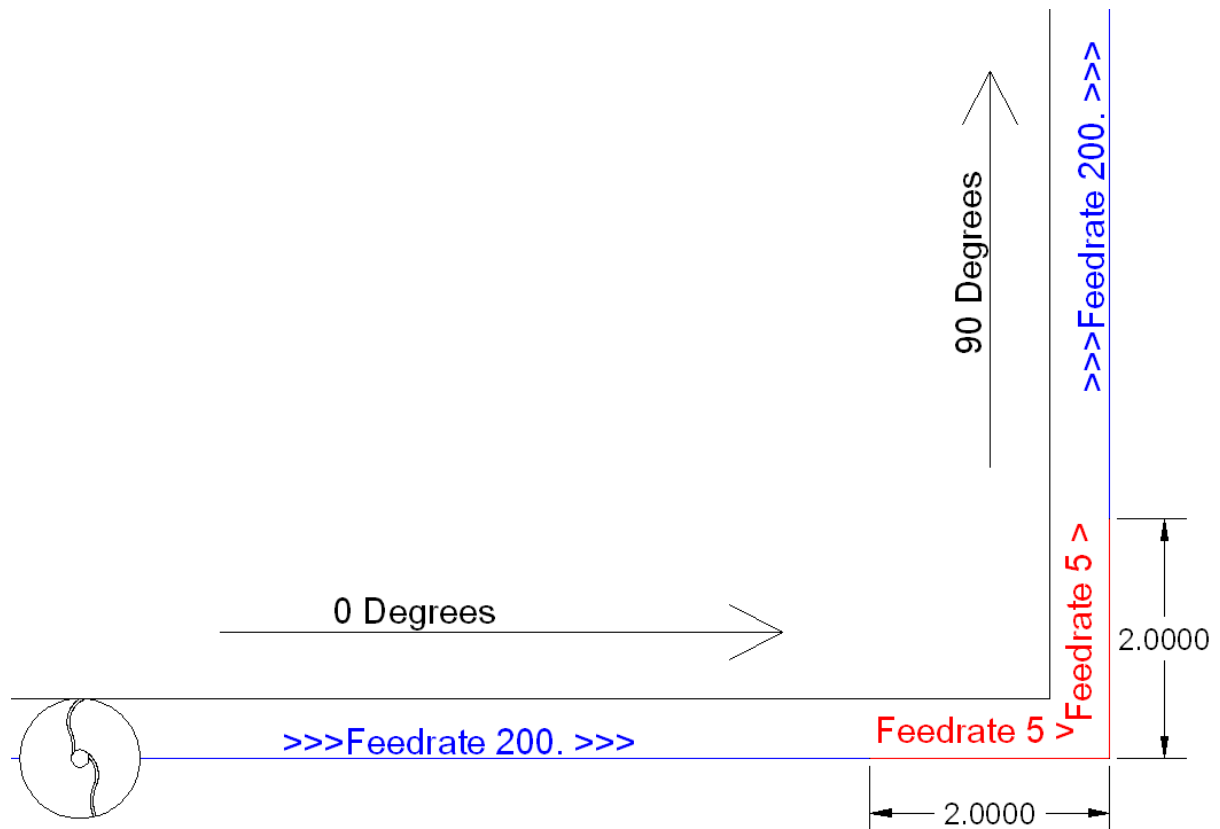
Each of these three formulas are the same.

The first element will be converted into two separate elements so that one can be the deceleration move. The first will be 8 units in length and the second (the deceleration segment) will be 2 units in length. The first element will have a feedrate of `_ACC_FEEDRATE` (200), which is the normal programmed feedrate.

The second element will have a feedrate of `_ACC_CLAMPMIN` (5).

This will allow the machine to ramp down from a feedrate of 200 to a feedrate of 5 over the course of 2 units, just prior to the corner.

Since the `_ACC_ACCDIST` is zero in this example, the third element (originally the second element at 90°) will then be broken up into 2 pieces equal to the last two on the deceleration side. The first will be 2 units long and have a feedrate of 5 and the second have a feedrate of 200 and the length of 8 units.



Example Code:

```
N01 G91 G01 X8.0 Y0.0 F200.
N02 G01 X2.0 Y0.0 F5.
N03 G01 X0.0 Y10.0 F200.
```

If the corner was filleted, the feedrate and length of the deceleration segment would be determined by using the radius feedrate scale ratio ($_ACC_CLAMP_RADIUS / _ACC_CLAMP_FEED$).

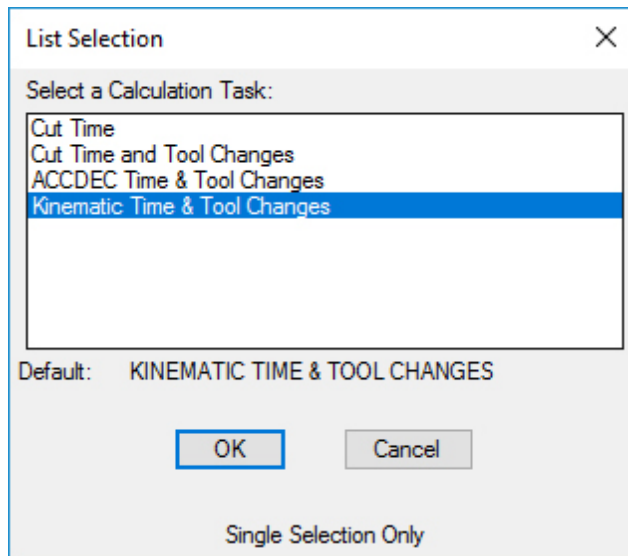
If $_ACC_GRAINDIR$ is not NIL, then additional limits would be placed on all feedrates that went on the cross grain angle based on the cross grain percentage. By multiplying the current feedrate by the cross grain percentage, a new feedrate is determined.

SPECIAL CONDITIONS:

If an element is not long enough to handle the required de-acceleration segment(dec-segment) length, then the feedrate will be applied to the start of the element. This will, at least, ramp down over the element's length even though full deceleration feedrate will not be achieved at the end of the element.

Perform Machine Calculations

After selecting, click on the 'Select' button to the right. A List Selection dialog window will appear. From this screen you have four choices:



Cut Time:

Selecting on this field, Router-CIM will figure the approximate cutting time, in minutes, for the Cut or Sequence selected.

Cut Time and Tool Changes:

Selecting on this field, Router-CIM will figure the approximate cutting time, in minutes, and will add the time for a tool change for the Cut or Sequence selected. The time figured for the tool change is in a NCVAR called $*TOOL_TIME*$. To change the value figured for the tool change:

- 1) Type in NCVAR at the command prompt.
- 2) Click on OTHER, scroll down and click on $*TOOL_TIME*$ and enter the new value.
- 3) Press <Enter>, then click on <OK>.

ACCDEC Time & Tool Change:

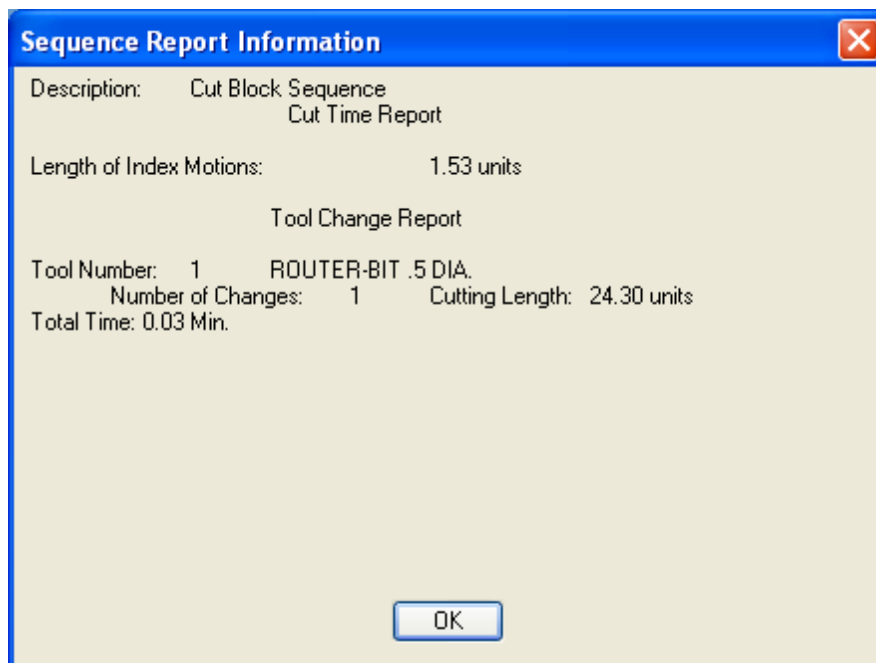
Selecting on this field, Router-CIM will figure the Acceleration/Deceleration, Cut Time and Tool Change Times for the Cut or Sequence selected. You will get a cut time based on acceleration and deceleration. This cut time is only valid if you make your code with ACCDEC on. If you did not use ACCDEC in your code, then select just Cut Time or Cut Time and Tool Changes.

Kinematic Time & Tool Change:

Selecting on this field, Router-CIM will figure the Cut Time and Tool Change Times based on the Kinematic Time Study for the Cut or Sequence selected.

For more information on the Kinematic Time Study Parameters, click [here](#).

NOTE: After running Sequence, a dialog window called Sequence Report Information will appear showing the time.



To see the report again after Sequence has been run, click on Sequence from the Control Panel or Router-CIM toolbar, click on the Clear All button at the top of the NC Sequence Builder screen, select Report Information, click on the Select button to the right, click on one of the two choices, click on <OK>, and then click on <OK>.

Using this option with Router Report turned on also will generate comments in the code like the ones shown below in red.

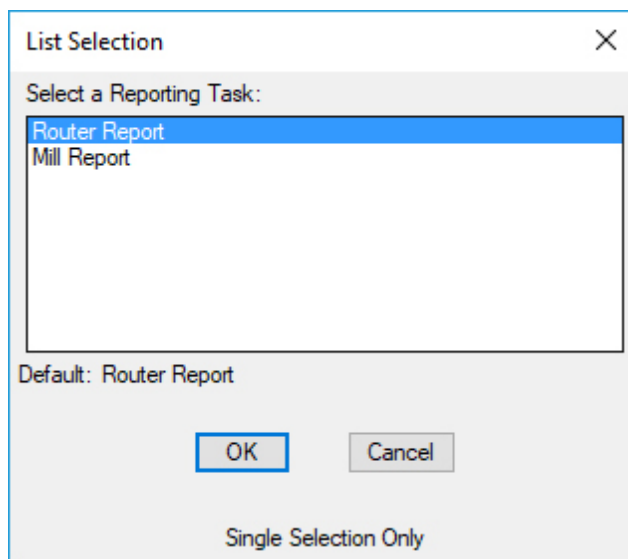
```

%
:12
(DRAWING NAME YOURDRAWINGNAME)
(PROGRAMMERS NAME HERE)
(11-06-2008 AT 11:03)
(ROUTER-BIT .5 DIA. TOOL 1 TOOL LENGTH 1 WORK CORD 55)
(CUTTER COMP 1)
(TOTAL TIME: 0.03 MIN.)
  
```

N1G00G17G20G28G40G80G91Z0M5
N2G90
N3G52X0Y0Z0

Report Information

After choosing this box, click on the Select button to the right. A List Selection dialog window will appear and from here you have two choices of output: Router Report or Mill Report. Under normal use you should choose Router Report.



Edit Tooling Motions

Edit Tool Motions provide the editing and emulation of selected tool paths

The Tool Path Editor provides for the displaying of information, emulation of tool path and the editing of tool paths. The three dialog interfaces are described as:

Tool Path Information - The main dialog screen is used to set modes and traverse the NC program. The display includes the NC Code and current settings that are in the NC Program. Selection modes are available to rapidly traverse through the NC Program. The modes for emulation are selected in this dialog.

Tool Path Emulation - This dialog provides for display settings and a traverse button to move through the NC Program during Emulation. The NC Program is emulated each time the NEXT button is activated. The mode of emulation was determined in the Information dialog display. Emulation methods include Tool Vector display, Tool Drop (inserting of tool block) and highlighting (drawing of the tool path in different colors).

Tool Path Editing - This dialog is used to pick the form of editing to apply and whether the edits are to be done globally or locally in the NC Program. When OK is selected, the actual edit takes place.

Tool Path Information

Positioning Modes
☒ Line/Arc/Text ☐ Path ☐ Cut

Positioning Information
 X = 6.5510 Y = -0.2490 Z = 0.2500

Post Commands: MCUSET

NC Code
 (MCULMS NAME^034ROUTER^034)
 (MCUSTOCK X0. Y0. Z0. OTL)
 G00G17G20G28G40G80G91Z0M5
 G90
 G52X0Y0Z0

Settings
 NC_INDEX
 SPEED/18000.00000
 SPINDLE/CW

Emulation
Display Mode
☐ Vector ☐ Tool Drop ☒ Highlight
Step Mode
☐ Single ☒ Continuous

OK Cancel Help Config... Emulate... Edit...

Areas in this dialog box that are not defined in other sections are described below:

Positioning Information

This area displays the current X,Y Z position at the location defined by the editor pointer. These coordinates at world space as defined by AutoCAD.

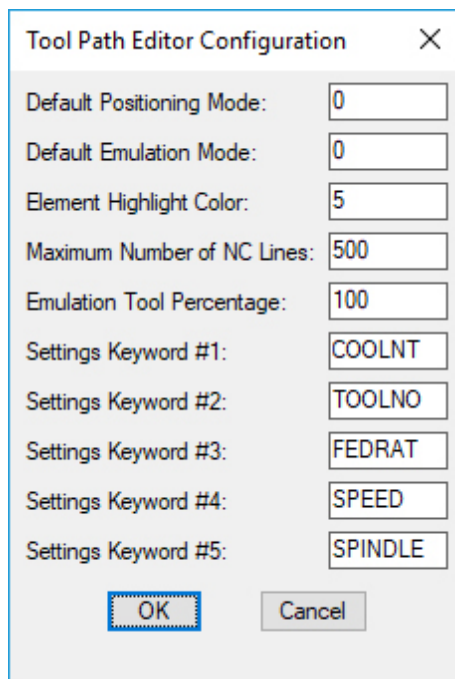
Post Commands

This area displays all the post processor commands used to develop the line of NC Code where the editor pointer is located.

NC Code List

This area displays the associated NC Code for the current Work Space. When the editor pointer is incremented the associated line of NC Code is selected in this list. This list can be used to move the editor pointer by scrolling to a location in the list and selecting the desired line of NC Code.

Settings



The image shows a 'Tool Path Editor Configuration' dialog box with a close button (X) in the top right corner. It contains several settings, each with a label and a text input field:

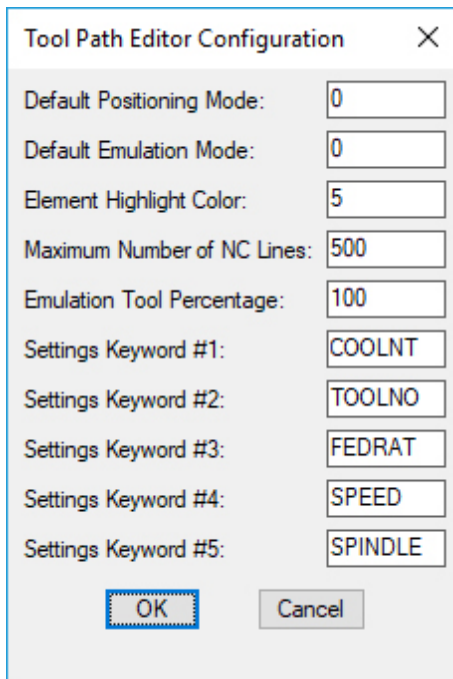
- Default Positioning Mode: 0
- Default Emulation Mode: 0
- Element Highlight Color: 5
- Maximum Number of NC Lines: 500
- Emulation Tool Percentage: 100
- Settings Keyword #1: COOLNT
- Settings Keyword #2: TOOLNO
- Settings Keyword #3: FEDRAT
- Settings Keyword #4: SPEED
- Settings Keyword #5: SPINDLE

At the bottom of the dialog are two buttons: 'OK' and 'Cancel'.

This area displays all the current settings defined in the Editor Configuration. Each time a Setting Keyword is encountered it is placed in this area of the dialog. Each time the Editor pointer is moved this scan is performed.

Sequence Emulator Options

The Sequence Emulator Options Set different modes and values that control the Tool Path Editor



Setting	Value
Default Positioning Mode:	0
Default Emulation Mode:	0
Element Highlight Color:	5
Maximum Number of NC Lines:	500
Emulation Tool Percentage:	100
Settings Keyword #1:	COOLNT
Settings Keyword #2:	TOOLNO
Settings Keyword #3:	FEDRAT
Settings Keyword #4:	SPEED
Settings Keyword #5:	SPINDLE

Default Positioning Mode

Positioning Modes: 0 = line/arc, 1 = path, 2 = cut

Line/Arc - positions along a cut by one single motion at a time.

Path - positions along a cut for an entire cut path at a time.

Cut - positions along an entire cut block at a time.

Default Emulation Mode

Emulation Modes: 0 = highlight, 1 = tool drop, 2 = vector

Highlight - draws the object using the Highlight color.

Tool Drop - inserts the tool geometry at intervals along the cut path.

Vector - draws vectors representing the tool along the cut path.

Element Highlight Color

Highlight Color - a number representing an AutoCAD color. An object is highlighted by redrawing the object using this color.

Maximum Number NC Lines

Maximum of NC Code Lines - a buffer of information is stored in memory in which the editor works on. The size of this work space is determined by the approximate number of NC code lines that would be generated within the work space.

Emulation Tool Percentage

Emulation Tool Percentage - When using Vector or Tool Drop emulation, the spacing between each emulation motion is determined by the percentage of the tool radius.

Settings Keyword #1

A postprocessor command word that will appear in the setting areas when encountered in the sequence.

Settings Keyword #2

A postprocessor command word that will appear in the setting areas when encountered in the sequence.

Settings Keyword #3

A postprocessor command word that will appear in the setting areas when encountered in the sequence.

Settings Keyword #4

A postprocessor command word that will appear in the setting areas when encountered in the sequence.

Settings Keyword #5

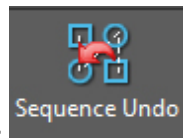
A postprocessor command word that will appear in the setting areas when encountered in the sequence.

Sequence Undo

Router-CIM Toolbar:



Router-CIM Ribbon:

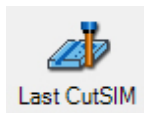


Keyboard: **SEQUNDO**

Upon completing of making code, you can undo the Sequence that was created by selecting the 'Sequence Undo' button. Once selected, simply select on any cut path or index line and the sequence will be undone and you will be back to your applied cut paths.

CutSIM in AutoCAD

Once your cut paths have been sequenced, you can access the CutSIM add-on directly from the Router-CIM Control panel.



Last CutSIM: This option will be grayed out initially until a sequence has been created. Once a sequence is created, this button will become active. When selected, it will automatically open the last .CL file that was created in the Router-CIM Interactive session.



CutSIM: This option will open the CutSIM program directly from the Control Panel. You will then be able to open any .CL that has been previously created.

Knowledge

Defining Knowledge

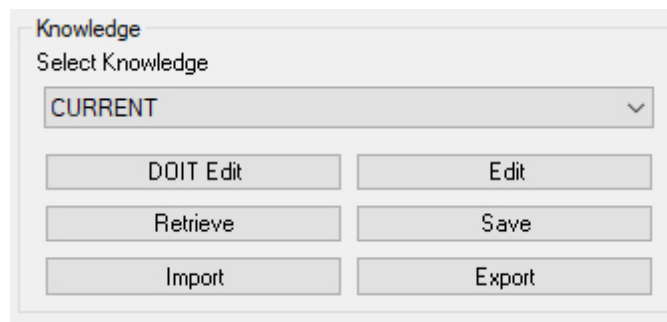
Router-CIM knowledge blocks are the most powerful part of the Router-CIM and Router-CIM system.

All of the information stored in Tool, Cycle, and Status Information (parameters) for *ANY* cut can be stored in a file, or, in a **knowledge** block. These cutting conditions can be stored and named within a drawing or stored and named on a hard or floppy disk. Disk based knowledge files can contain several knowledge blocks, making them a sort of library of cutting conditions.

You can store as many of these knowledge libraries on your computer as you wish and import them into a drawing when needed. Alternately, you may store several knowledges in your default drawing, and use them whenever Router-CIM or Router-CIM are running.

Router-CIM uses knowledge stored in individual drawings (in the knwdir folder) to allow you to use specific knowledge for a particular job. By keeping the knowledge for a specific function sperate, you keep the file sizes smaller, and the system runs faster.

Different forms of the knowledge command exist for building, extracting, and saving cutting conditions within a drawing or from a disk drive.



Retrieve Knowledge

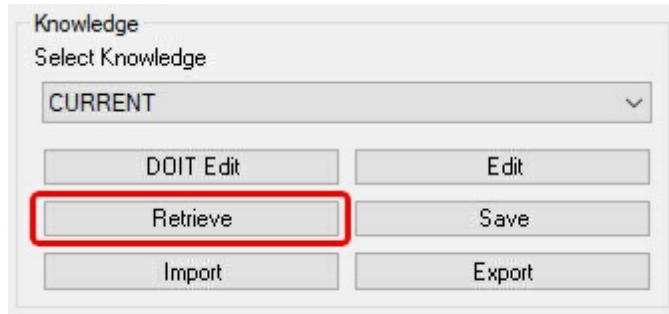
Router-CIM Toolbar: 



Router-CIM Ribbon:

Keyboard: K

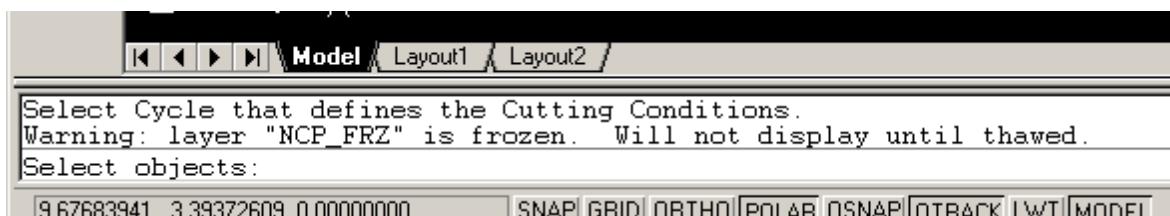
Control Panel:



The Retrieve Knowledge command is used to reset cutting conditions from an existing Cut. All the cuts that exist in a Router-CIM drawing have their knowledge stored in the drawing for retrieval.

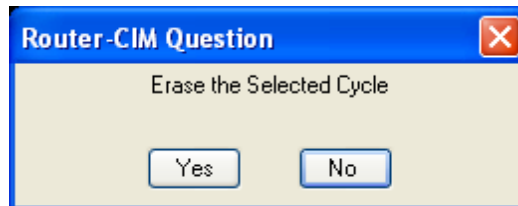
Retrieve

Selecting retrieve knowledge will first prompt you to Select Cycle that defines the Cutting Conditions that you want to retrieve.

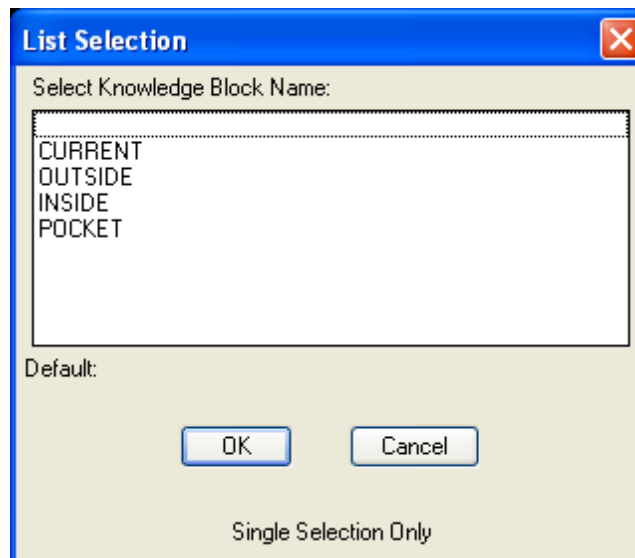


If there is a cut on the screen that you want the knowledge from, select it. You will then be prompted as to whether or not you want to erase the selected cut. If you wish to make edits to the cut for current use, answer Yes. Otherwise answer No. If you select **No**, then the Control Panel will appear showing the attribute values for that condition - you can change any of the attributes or even change the tool or cycle and use any part of the condition in a new Cut.

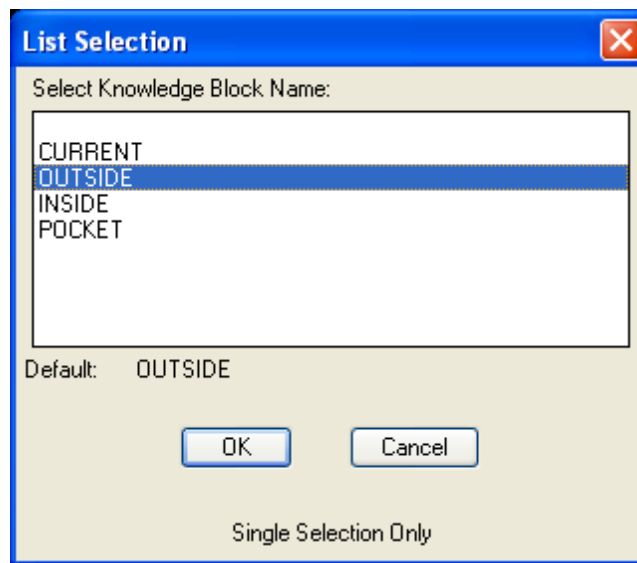
If you answer **Yes** to the erase prompt, your Cut will be erased but the attribute values for that condition will still appear in the Control Panel.



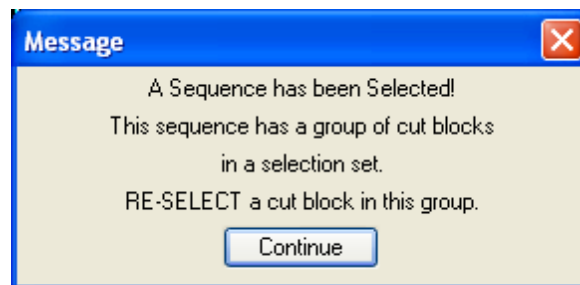
If there is no cut on the screen and you want to select the knowledge from a list of available knowledges, press <ENTER> and a list will be shown of available knowledges.



Select the one that you want the information from.



NOTE: If you use *Retrieve Knowledge* and select a *Sequence*, you will see a message that a *Sequence* is selected and be asked to select a *CUT* block. If you select *Continue*, and select the cut again, you will get the knowledge from the cut and the *Sequence* will still exist.



Save Knowledge

Router-CIM Toolbar:

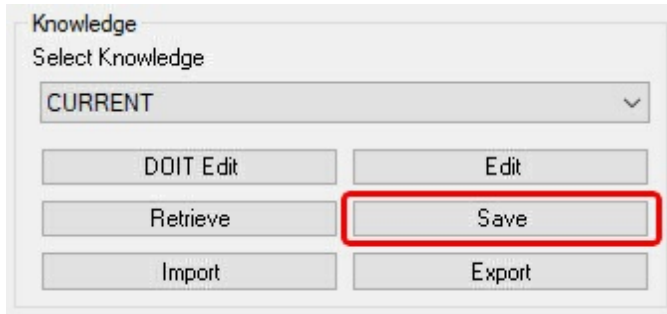




Router-CIM Ribbon:

Keyboard: **SK**

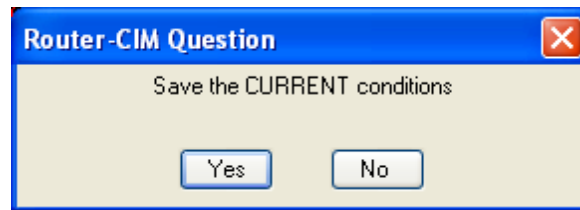
Control Panel:



The Save Knowledge command is used to Save and name cutting conditions within a drawing.

How to Save a Knowledge

- Select the Save Knowledge button (shown above), enter **SK** from the keyboard, or use the Save button on the Control Panel to start the command.
- You are prompted to select a Cut cycle to Save. If you select a CUT that you want to save, you are then prompted to enter a knowledge block name (29 character maximum).



- You may want to only save the conditions on the Control Panel, and you don't have a cut on the screen to select. When you are prompted to select a Cut cycle to Save, press **Enter** and you will be prompted to Save the current cutting condition (screen above). Click on '**YES**' and you will be asked to give that knowledge a name.

```
Command: SK
Select Cycle that defines the Cutting Conditions
Select objects:
Enter Knowledge Block Name: MYKNOWLEDGE
```

- Enter a name and the knowledge is saved. In this case the name of the knowledge was MYKNOWLEDGE.

```
Enter Knowledge Block Name: MYKNOWLEDGE
Developing Knowledge Block...
MYKNOWLEDGE Knowledge Block Stored.
Command: |
```

NOTE: When you save knowledge, it is only saved in the current drawing. In order to make it available to other drawings, you must Export the knowledge. You can store several individual cut knowledges in one large disk knowledge file. This makes them easier to store and retrieve.

NOTE: Use the Retrieve Knowledge ("K" at the keyboard) command to retrieve the Saved Knowledge (See [Retrieve Knowledge](#)).

Exporting Knowledge

Router-CIM Toolbar:

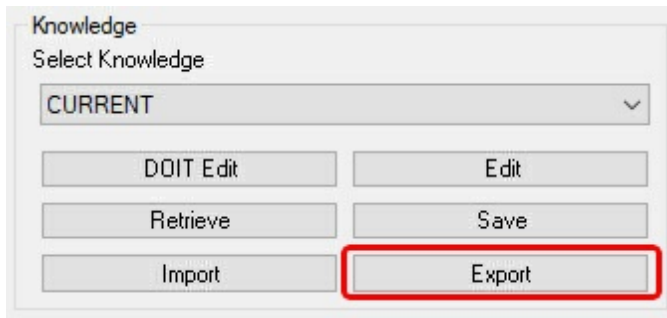




Router-CIM Ribbon:

Keyboard: EK

Control Panel:

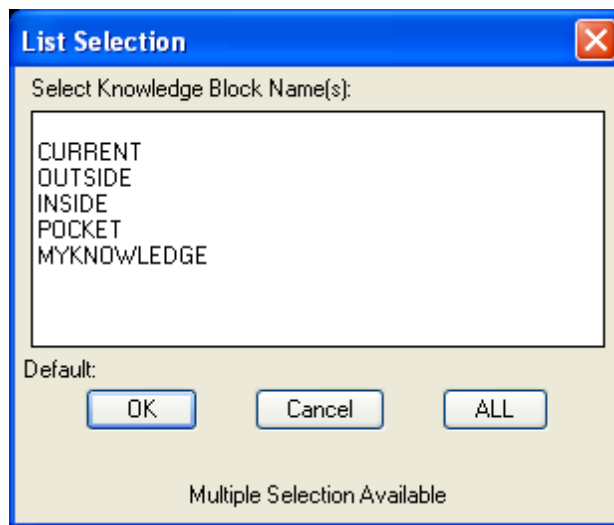


The Export Knowledge command is used to Save and name cutting conditions to a disk drive. Use the Save Knowledge command first because you must Save the knowledge before you can Export it.

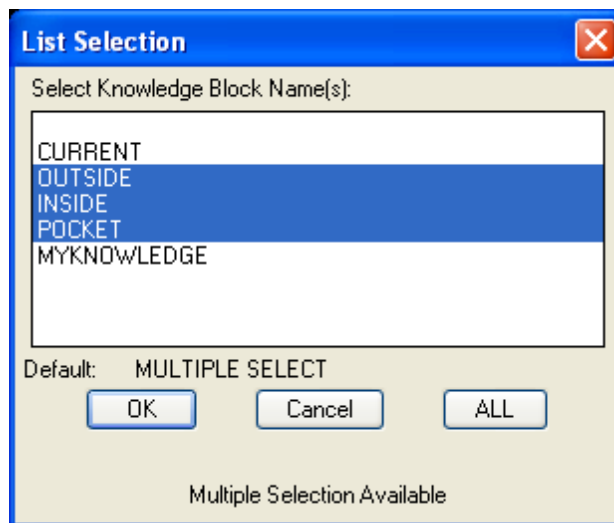
Export

In order to Export Knowledge the following steps must occur:

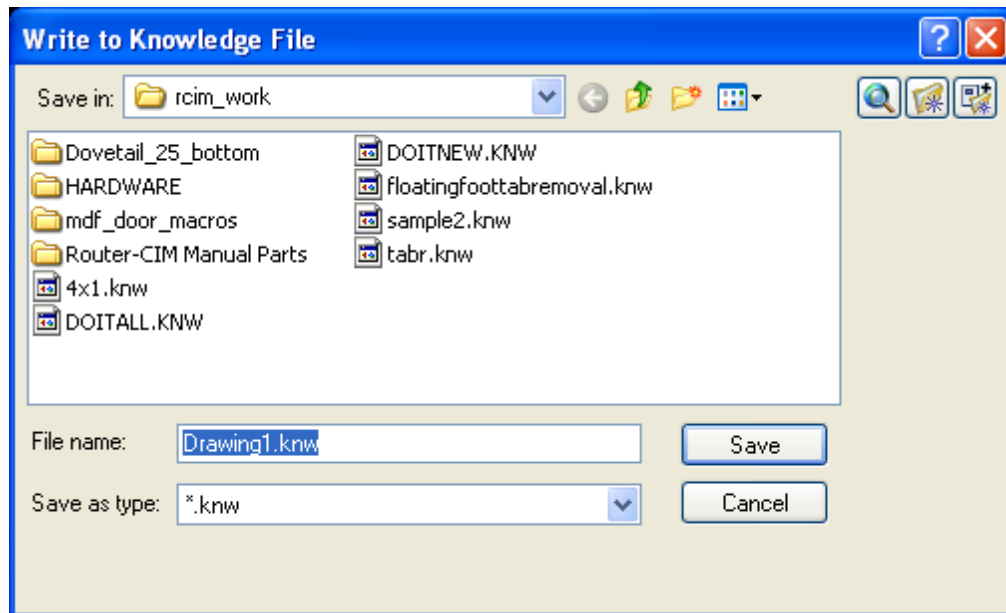
Select the **Export Knowledge** button, and a pop up menu showing the currently saved cutting conditions will appear.



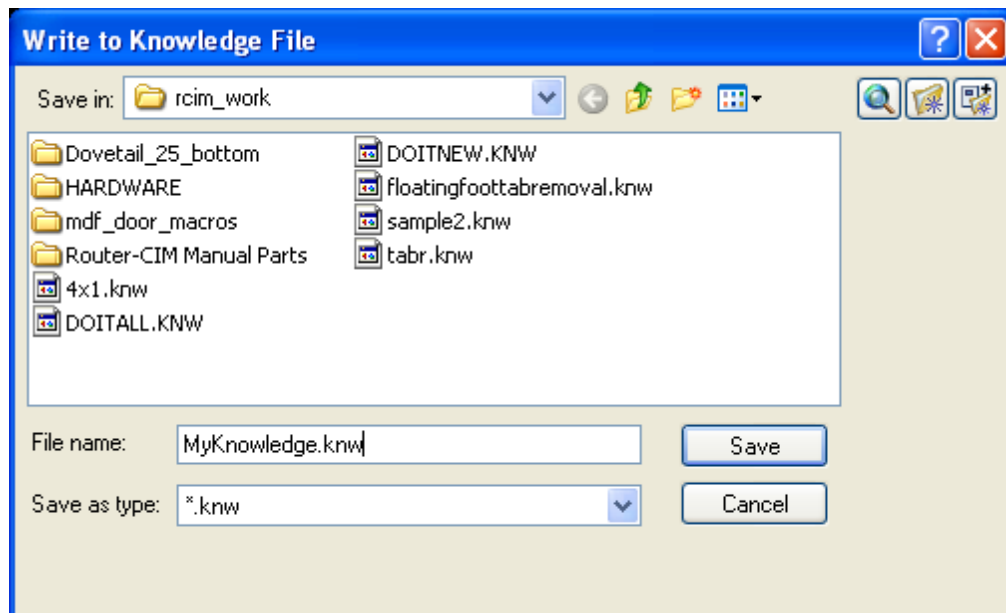
- Pick the cutting condition(s) to Export. Select **only** the ones you want to export. Selecting the blank line at the top here and the Current knowledge may cause issues because they are not a name you will want to retrieve later but they will still show up in the list.



Multiple selection is available by selecting the first knowledge, holding the Shift Key, and selecting the last one. The selection above illustrates this.



- You will be shown a window to enter an Export File Name (above). The default name will be shown as the drawing name.



- If desired, enter a different filename.

```
Knowledge Selection - Examining Existing knowledge...
Exporting Knowledge...
Selected Knowledge Stored in the C:\RCIM_WORK\MYKNOWLEDGE.KNW file.
Command:
```

- The Exported file is stored on the disk drive as a .KNW file. The prompt above should appear in your AutoCAD command line.

Importing Knowledge

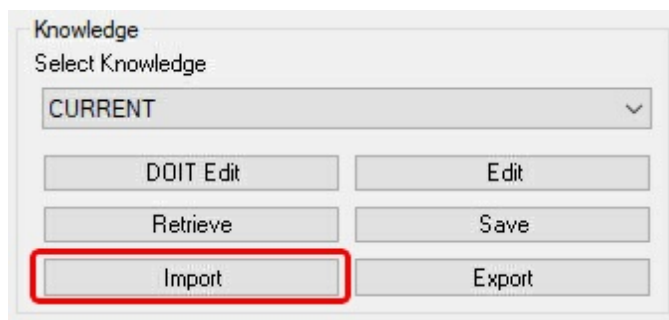
Router-CIM Toolbar:



Router-CIM Ribbon:

Keyboard: IK

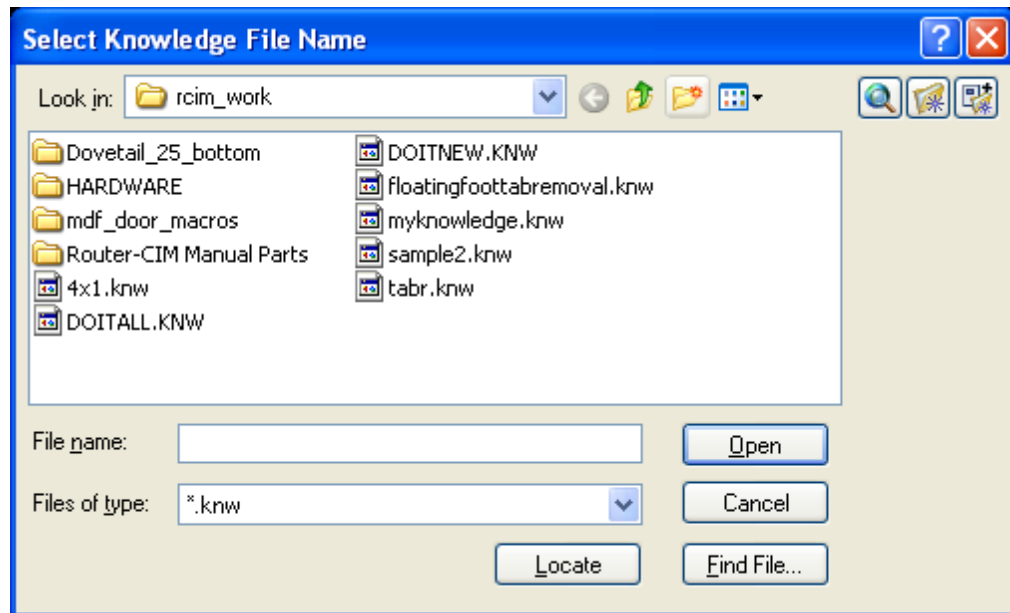
Control Panel:



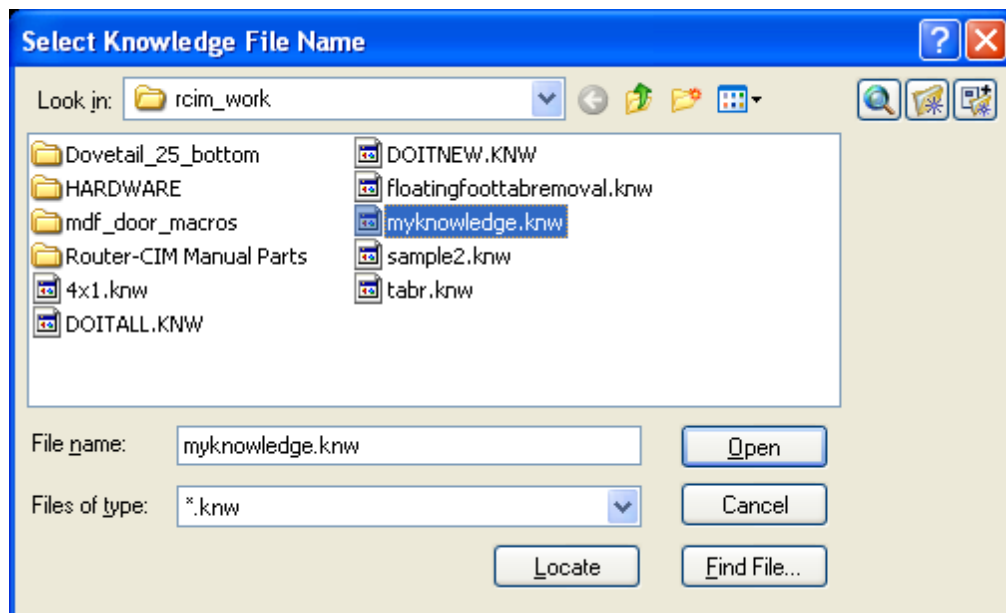
The Import Knowledge command will insert cutting conditions from a disk drive knowledge file.

Import

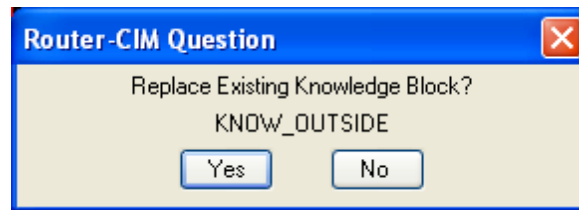
Select Import Knowledge and you will be prompted to enter the name of the knowledge block to Import.



- You may then select the knowledge file that you wish to import into the current drawing.



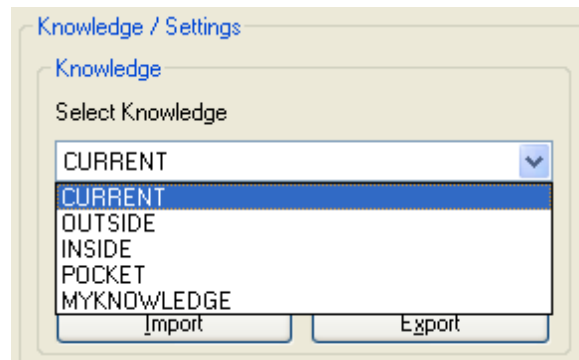
- Once selected, press the '**Open**' button.
- If the knowledge file that you are importing contains cut knowledges that are already in the drawing that you are using, you will be prompted to overwrite the existing knowledges with the ones from disk.



- Selecting **'Yes'** replaces the ones in the drawing with the ones from the file you are importing. Selecting No leaves the knowledge from the current drawing intact and the duplicate is not imported.
- After the knowledges have been imported the following message should appear at the command prompt.

```
Knowledge Selection - Examining Existing knowledge...
Importing Knowledge...
C:\RCIM_WORK\MYKNOWLEDGE.KNW Knowledge File Imported.
Command:
```

- The cut knowledges imported can now be seen in the Knowledge pull down menu on the control panel.



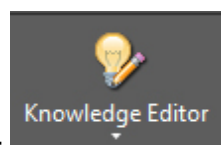
If you put multiple cutting conditions into a single Export file, the Import Knowledge command will insert all of the individual cutting conditions at one time. In the case above, there were several knowledges imported.

Edit Knowledge

Router-CIM Toolbar:

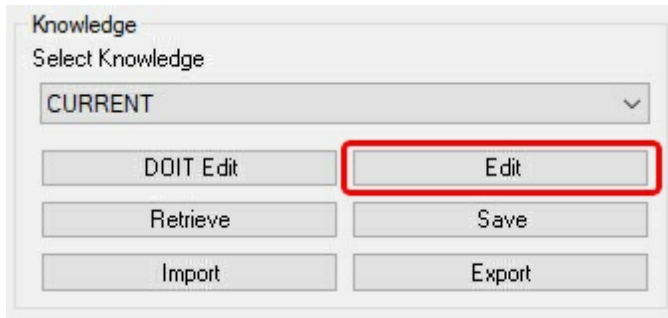


Router-CIM Ribbon:



Keyboard: KE

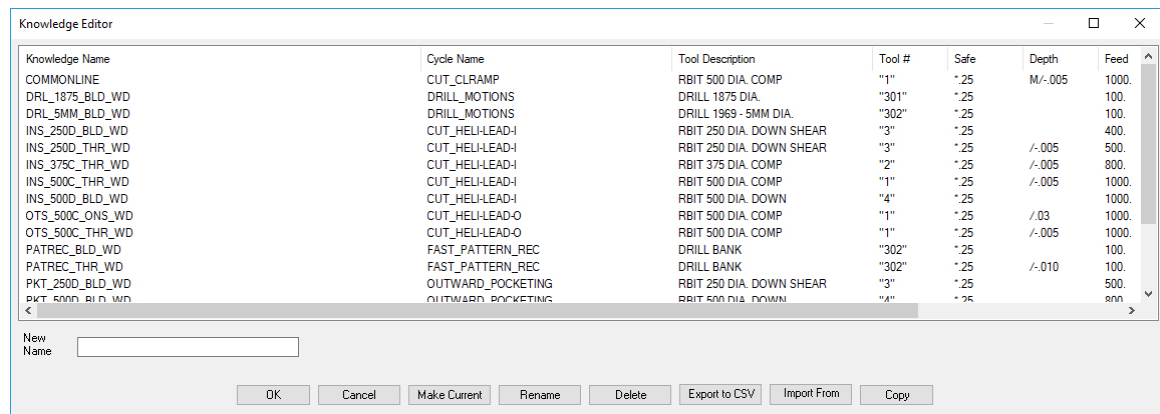
Control Panel:



Editing a Selected Knowledge

Selecting this icon from the Router-CIM toolbar will bring you to the interface below.

This displays the Knowledge Editor.

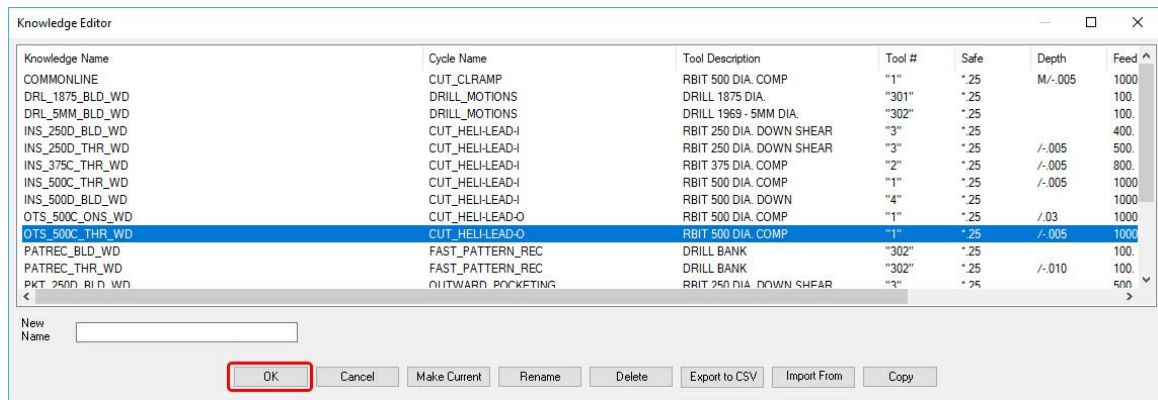


Using the Knowledge Editor

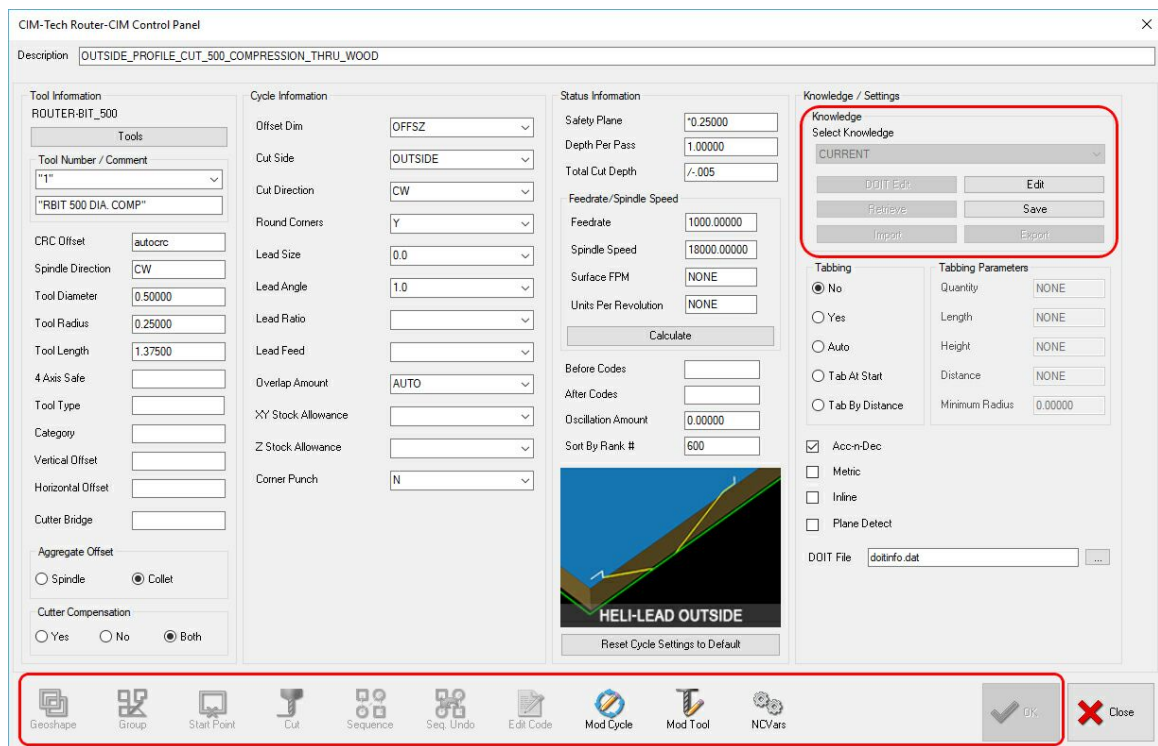
Knowledge Quick Edit

The Knowledge Quick Edit will allow you to change certain parameters of the selected knowledge and re-save it to the list without changing its name. This has the benefit of allowing you to change items like a tool number or tool, and then be back in the Knowledge Editor with those changes saved.

- Select a knowledge from the list so that it is highlighted.



- Now select the **'OK'** button.
- The Control Panel is displayed, with only certain items available to you. You can change a tool, tool description, cycle, cycle parameters. You cannot Geoshape, Start Point Edit, even Cut is not allowed.



- Change the tool to a 1" end mill and the description, and click on **'Save'** in the Knowledge section, and you are returned to the Knowledge Editor with your changes intact.

Knowledge Editor

Knowledge Name	Cycle Name	Tool Description	Tool #	Safe	Depth	Feed
OTS_500C_THR_WD	CUT_HELI-LEAD-O	ROUTER-BIT 1.00 DIA.	"2"	*0.25000	/-.005	1000
INS_500C_THR_WD	CUT_HELI-LEAD-I	RBIT 500 DIA. COMP	"1"	*.25	/-.005	1000
INS_250D_BLD_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25		400
DRL_1875_BLD_WD	DRILL_MOTIONS	DRILL 1875 DIA.	"301"	*.25		100
DRL_5MM_BLD_WD	DRILL_MOTIONS	DRILL 1969 - 5MM DIA.	"302"	*.25		100
SCRAP	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000
SKELETON	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000
PKT_500D_BLD_WD	OUTWARD_POCKETING	RBIT 500 DIA. DOWN	"4"	*.25		800
INS_375C_THR_WD	CUT_HELI-LEAD-I	RBIT 375 DIA. COMP	"2"	*.25	/-.005	800
INS_250D_THR_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25	/-.005	500
PATREC_BLD_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25		100
PATREC_THR_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25	/-.010	100
PKT_250D_BLD_WD	OUTWARD_POCKETING	RBIT 250 DIA. DOWN SHEAR	"3"	*.25		500

New Name:

OK Cancel Make Current Rename Delete Export to CSV Import From Copy

- When no Knowledges are selected, click on 'OK' to leave the Knowledge Editor.

Making a Knowledge Current

- To make a Knowledge current in the Status Page, select it from the list so that it is highlighted.

Knowledge Editor

Knowledge Name	Cycle Name	Tool Description	Tool #	Safe	Depth	Feed
OTS_500C_THR_WD	CUT_HELI-LEAD-O	RBIT 500 DIA. COMP	"1"	*.25	/-.005	1000
INS_500C_THR_WD	CUT_HELI-LEAD-I	RBIT 500 DIA. COMP	"1"	*.25	/-.005	1000
INS_250D_BLD_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25		400
DRL_1875_BLD_WD	DRILL_MOTIONS	DRILL 1875 DIA.	"301"	*.25		100
DRL_5MM_BLD_WD	DRILL_MOTIONS	DRILL 1969 - 5MM DIA.	"302"	*.25		100
SCRAP	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000
SKELETON	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000
STAYDOWN	CUT_CHANNEL	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000
PKT_500D_BLD_WD	OUTWARD_POCKETING	RBIT 500 DIA. DOWN	"4"	*.25		800
INS_375C_THR_WD	CUT_HELI-LEAD-I	RBIT 375 DIA. COMP	"2"	*.25	/-.005	800
INS_250D_THR_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25	/-.005	500
PATREC_BLD_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25		100
PATREC_THR_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25	/-.010	100

New Name:

OK Cancel **Make Current** Rename Delete Export to CSV Import From Copy

- Then select 'Make Current', and the knowledge will be shown on the Control Panel.

CIM-Tech Router-CIM Control Panel

Description: OUTSIDE_PROFILE_CUT_500_COMPRESSION_THRU_WOOD

Tool Information
ROUTER-BIT_500

Tools

Tool Number / Comment
"1"

"RBIT 500 DIA. COMP"

CRC Offset: autocrc

Spindle Direction: CW

Tool Diameter: 0.50000

Tool Radius: 0.25000

Tool Length: 1.37500

4 Axis Safe:

Tool Type:

Category:

Vertical Offset:

Horizontal Offset:

Cutter Bridge:

Aggregate Offset
☐ Spindle ☒ Collet

Cutter Compensation
☐ Yes ☐ No ☒ Both

Cycle Information

Offset Dim: OFFSZ

Cut Side: OUTSIDE

Cut Direction: CW

Round Corners: Y

Lead Size: 0.0

Lead Angle: 1.0

Lead Ratio:

Lead Feed:

Overlap Amount: AUTO

XY Stock Allowance:

Z Stock Allowance:

Corner Punch: N

Status Information

Safety Plane: 0.25000

Depth Per Pass: 1.00000

Total Cut Depth: +/- .005

Feederate/Spindle Speed

Feederate: 1000.00000

Spindle Speed: 18000.00000

Surface FPM: NONE

Units Per Revolution: NONE

Calculate

Before Codes:

After Codes:

Oscillation Amount: 0.00000

Sort By Rank #: 600

Reset Cycle Settings to Default

Knowledge / Settings

Knowledge
Select Knowledge
CURRENT

DOIT Edit Edit

Retrieve Save

Import Export

Tabbing
☒ No
☐ Yes
☐ Auto
☐ Tab At Start
☐ Tab By Distance

Tabbing Parameters

Quantity: NONE

Length: NONE

Height: NONE

Distance: NONE

Minimum Radius: 0.00000

☒ Acc-m-Dec
☐ Metric
☐ Inline
☐ Plane Detect

DOIT File: doitinfo.dat

OK Close

Rename a Knowledge

- To Rename a Knowledge, first select it in the window to make it current, then enter the new name in the appropriate box at the bottom left of the window.

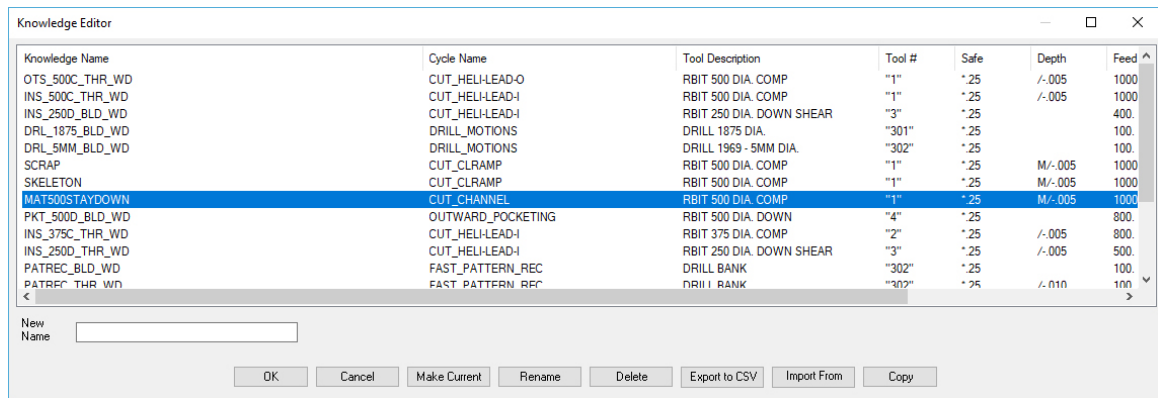
Knowledge Editor

Knowledge Name	Cycle Name	Tool Description	Tool #	Safe	Depth	Feed
SKELETON	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/- .005	1000.
STAYDOWN	CUT_CHANNEL	RBIT 500 DIA. COMP	"1"	*.25	M/- .005	1000.
PKT_500D_BLD_WD	OUTWARD_POCKETING	RBIT 500 DIA. DOWN	"4"	*.25		800.
INS_375C_THR_WD	CUT_HELI-LEAD-I	RBIT 375 DIA. COMP	"2"	*.25	/-.005	800.
INS_250D_THR_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25	/-.005	500.
PATREC_BLD_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25		100.
PATREC_THR_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25	/-.010	100.
PKT_250D_BLD_WD	OUTWARD_POCKETING	RBIT 250 DIA. DOWN SHEAR	"3"	*.25		500.
INS_500D_BLD_WD	CUT_HELI-LEAD-I	RBIT 500 DIA. DOWN	"4"	*.25		1000.
OTS_500C_ONS_WD	CUT_HELI-LEAD-O	RBIT 500 DIA. COMP	"1"	*.25	/-.03	1000.
COMMONLINE	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/- .005	1000.
ROUND	CUT_SPOIL_CEN	ROUND OVER 1.00 DIA. .125 RAD.	"1"	*0.25		350.

New Name: MAT500STAYDOWN

OK Cancel Make Current **Rename** Delete Export to CSV Import From Copy

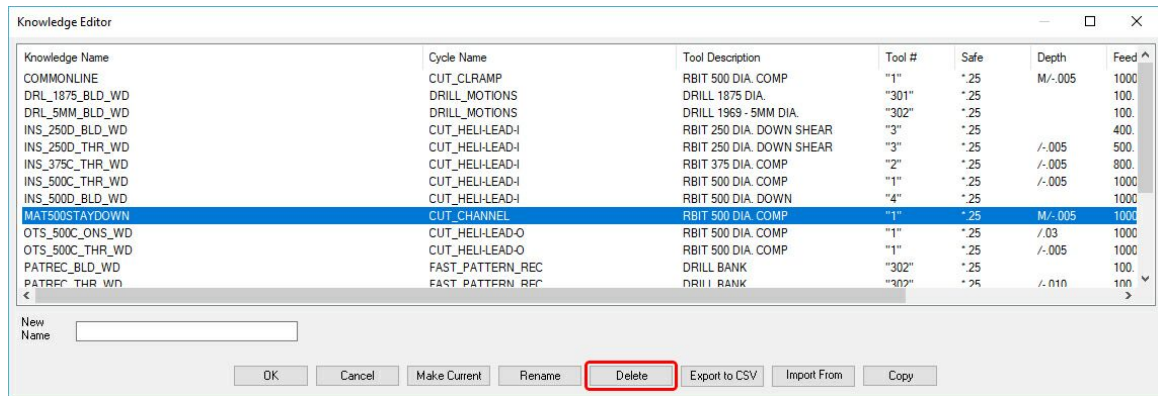
- Click on the '**Rename**' button



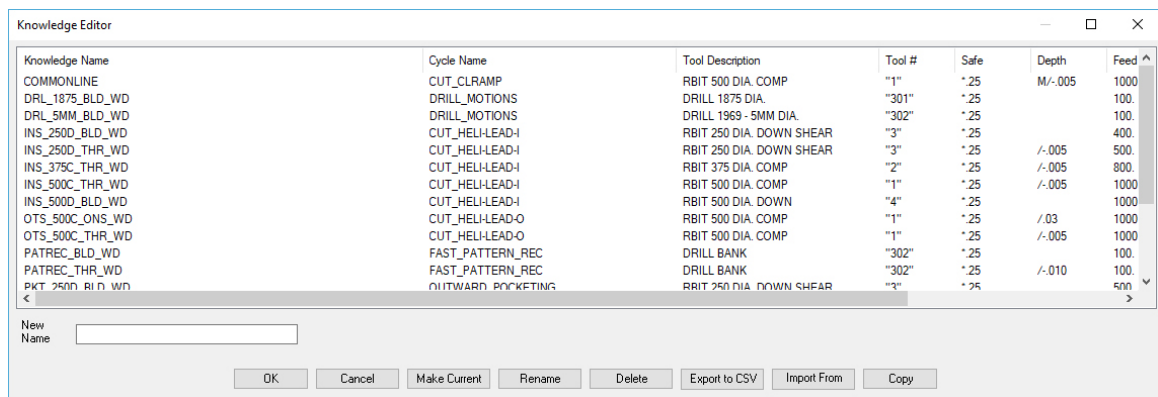
- The knowledge now appears in the list under the new name.

Deleting a Knowledge

- Delete a Knowledge by selecting it in the window to make it current and then click on the **'Delete'** button.

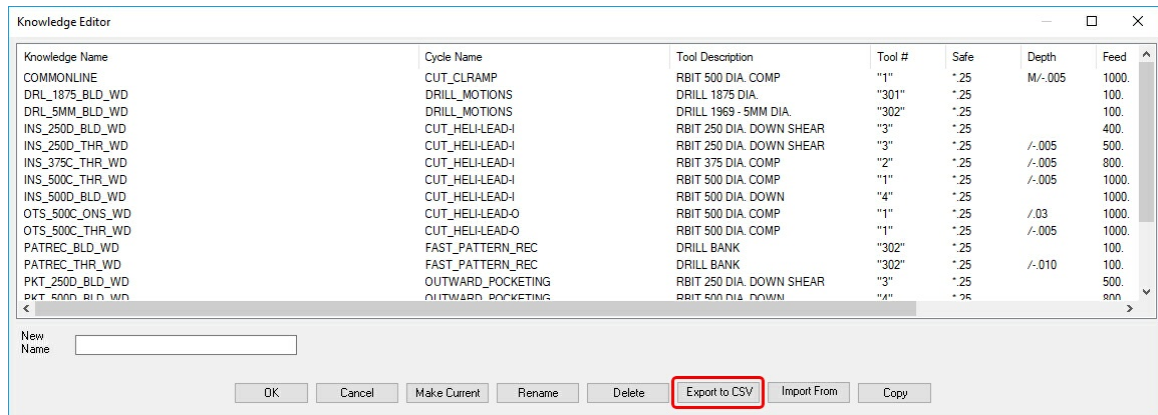


- The Knowledge is deleted from the available selections.

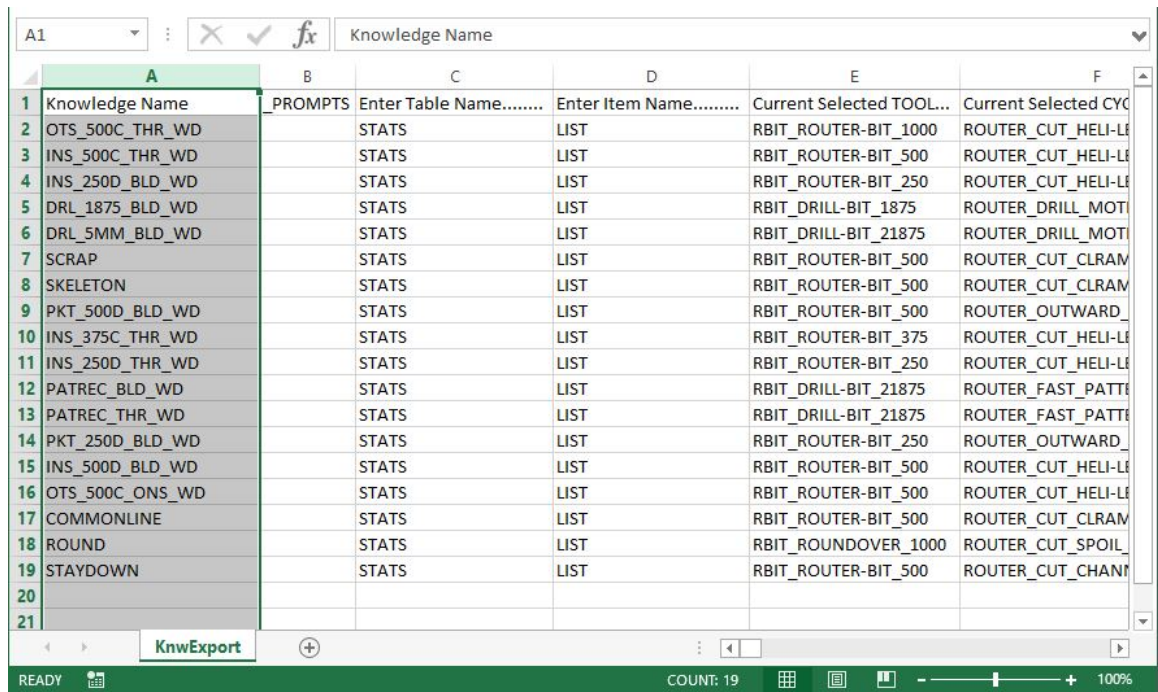


Export to CSV

The Export to CSV will create a comma delimited file that can be used as a back-up or import into another Knowledge drawing.



You will be prompted for a name and a save location and then the CSV will be created:



Import from CSV

The Import from CSV will bring in a comma delimited that was created by the Export to CSV option.

Knowledge Name	Cycle Name	Tool Description	Tool #	Safe	Depth	Feed	Speed	Diameter
OTS_500C_THR_WD	CUT_HELI-LEAD-O	ROUTER-BIT 1.00 DIA.	"2"	*.25000	/.005	1000.00000	18000.00000	1.00000
INS_500C_THR_WD	CUT_HELI-LEAD-I	RBIT 500 DIA. COMP	"1"	*.25	/.005	1000.	18000	0.50000
INS_250D_BLD_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25		400.	18000	0.25000
DRL_1875_BLD_WD	DRILL_MOTIONS	DRILL 1875 DIA.	"301"	*.25		100.	18000	0.18750
DRL_5MM_BLD_WD	DRILL_MOTIONS	DRILL 1969 - 5MM DIA.	"302"	*.25		100.	18000	0.19690
SCRAP	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000.	18000	0.50000
SKELETON	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000.	18000	0.50000
PKT_500D_BLD_WD	OUTWARD_POCKETING	RBIT 500 DIA. DOWN	"4"	*.25		800.	18000	0.50000
INS_375C_THR_WD	CUT_HELI-LEAD-I	RBIT 375 DIA. COMP	"2"	*.25	/.005	800.	18000	0.37500
INS_250D_THR_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25	/.005	500.	18000	0.25000
PATREC_BLD_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25		100.	18000	0.19690
PATREC_THR_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25	/.010	100.	18000	0.19690
PKT_250D_BLD_WD	OUTWARD_POCKETING	RBIT 250 DIA. DOWN SHEAR	"3"	*.25		500.	18000	0.25000

New Name:

OK Cancel Make Current Rename Delete Export to CSV **Import From CSV** Copy

Copy a Knowledge

To copy a selected knowledge, highlight the knowledge that you want to copy, type in a new name for the knowledge in the 'New Name' field and select the 'Copy' button.

Knowledge Name	Cycle Name	Tool Description	Tool #	Safe	Depth	Feed
OTS_500C_THR_WD	CUT_HELI-LEAD-O	ROUTER-BIT 1.00 DIA.	"2"	*.25000	/.005	1000.
INS_500C_THR_WD	CUT_HELI-LEAD-I	RBIT 500 DIA. COMP	"1"	*.25	/.005	1000.
INS_250D_BLD_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25		400.
DRL_1875_BLD_WD	DRILL_MOTIONS	DRILL 1875 DIA.	"301"	*.25		100.
DRL_5MM_BLD_WD	DRILL_MOTIONS	DRILL 1969 - 5MM DIA.	"302"	*.25		100.
SCRAP	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000.
SKELETON	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000.
PKT_500D_BLD_WD	OUTWARD_POCKETING	RBIT 500 DIA. DOWN	"4"	*.25		800.
INS_375C_THR_WD	CUT_HELI-LEAD-I	RBIT 375 DIA. COMP	"2"	*.25	/.005	800.
INS_250D_THR_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25	/.005	500.
PATREC_BLD_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25		100.
PATREC_THR_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25	/.010	100.
PKT_250D_BLD_WD	OUTWARD_POCKETING	RBIT 250 DIA. DOWN SHEAR	"3"	*.25		500.

New Name:

OK Cancel Make Current Rename Delete Export to CSV Import From **Copy**

The copied knowledge will now appear in the Knowledge Name column available for use or editing.

Knowledge Name	Cycle Name	Tool Description	Tool #	Safe	Depth	Feed
COMMONLINE	CUT_CLAMP	RBIT 500 DIA. COMP	"1"	*.25	M/-.005	1000.
DRL_1875_BLD_WD	DRILL_MOTIONS	DRILL 1875 DIA.	"301"	*.25		100.
DRL_5MM_BLD_WD	DRILL_MOTIONS	DRILL 1969 - 5MM DIA.	"302"	*.25		100.
INS_250D_BLD_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25		400.
INS_250D_THR_WD	CUT_HELI-LEAD-I	RBIT 250 DIA. DOWN SHEAR	"3"	*.25	/.005	500.
INS_375C_THR_WD	CUT_HELI-LEAD-I	RBIT 375 DIA. COMP	"2"	*.25	/.005	800.
INS_500C_THR_WD	CUT_HELI-LEAD-I	RBIT 500 DIA. COMP	"1"	*.25	/.005	1000.
INS_500D_BLD_WD	CUT_HELI-LEAD-O	RBIT 500 DIA. DOWN	"4"	*.25		1000.
OTS_500C_ONS_WD	CUT_HELI-LEAD-O	ROUTER-BIT 1.00 DIA.	"2"	*.25000	/.005	1000.
OTS_500C_THR_WD	CUT_HELI-LEAD-O	ROUTER-BIT 1.00 DIA.	"2"	*.25000	/.005	1000.
OTS_500C_THR_WD_COPY	CUT_HELI-LEAD-O	ROUTER-BIT 1.00 DIA.	"2"	*.25000	/.005	1000.
PATREC_BLD_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25		100.
PATREC_THR_WD	FAST_PATTERN_REC	DRILL BANK	"302"	*.25	/.010	100.

New Name:

OK Cancel Make Current Rename Delete Export to CSV Import From Copy

Group Knowledge

Router-CIM Toolbar:





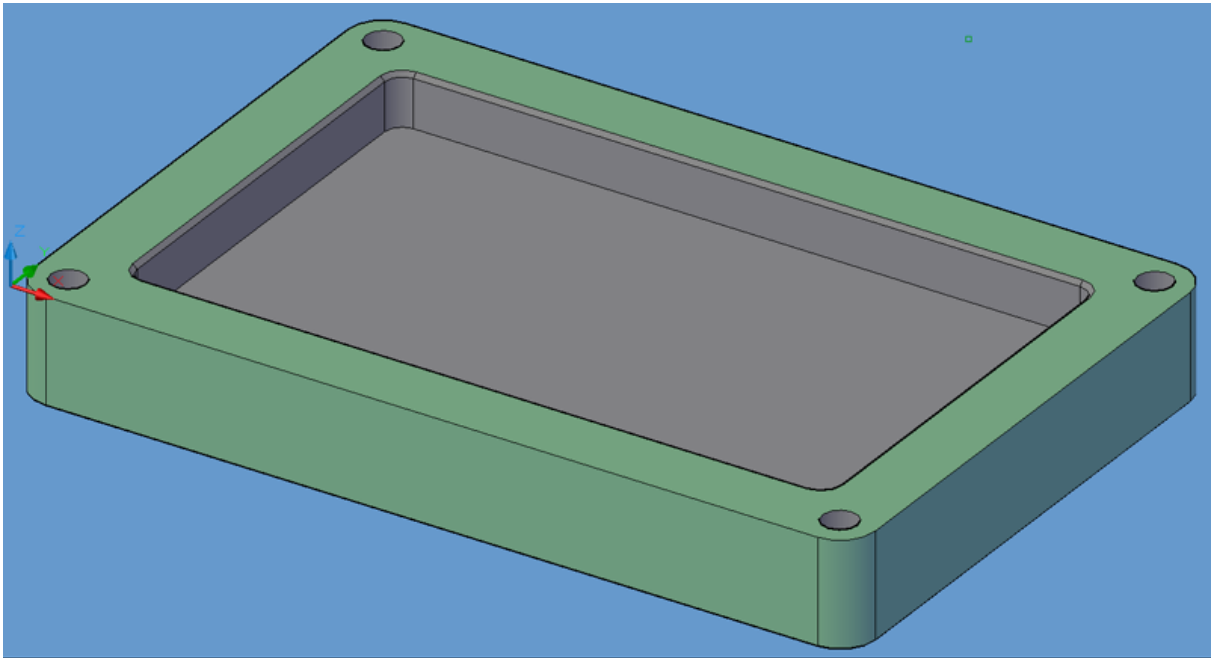
Router-CIM Ribbon:

Keyboard: **GK**

The Group Knowledge command will allow you to group several cutting conditions into one knowledge block so that all of the operations occur, in order, on a given shape by using one knowledge selection. If, for instance, you wanted to create a pocket of any given size with a roughing tool, finish with a different tool, then chamfer the edges, this can now be completed with a single selection using Group Knowledge.

You can only select Group Knowledge from the Toolbar, the pull-down menu, or typing "GK" on the keyboard. There is no selection from the Control Panel.

Consider the following part.

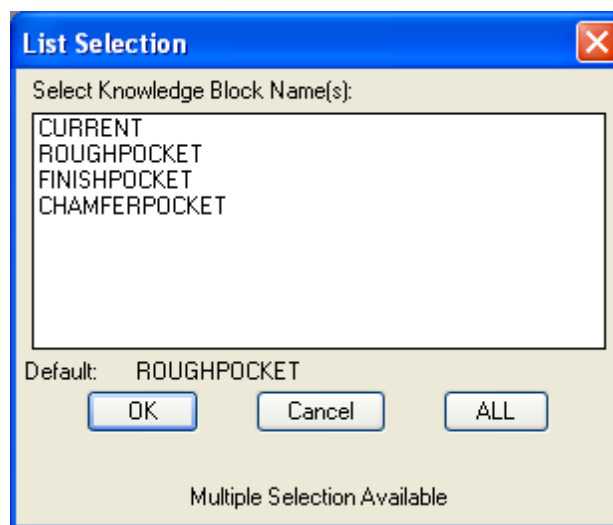


To complete this pocket with the specifications listed above, we need three knowledges. One to rough the pocket, one to finish the pocket, and one to make the chamfer along the top edge of the pocket.

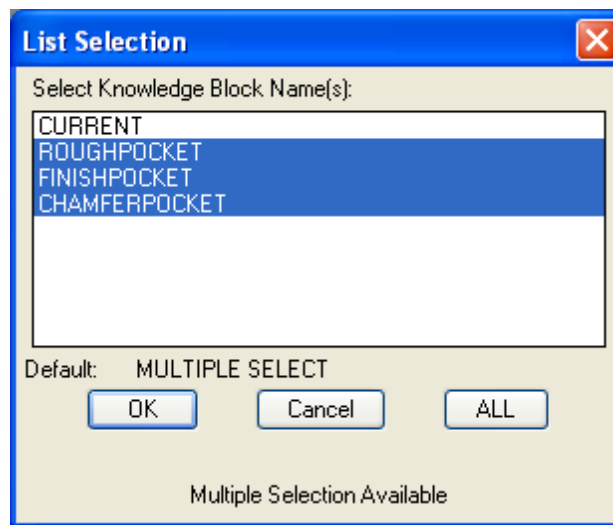
While these could be completed in three operations, if you had several parts to make in this fashion, and they had pockets of varying geometry, then a Group Knowledge is easier to use per part.

Making a Group Knowledge

- Select '**Group Knowledge**' from the toolbar. A window will appear with the current knowledges contained in the current drawing.



- Select the cuts **IN ORDER** for the Group Knowledge. This is the order that they will cut and sequence in.



- Select '**OK**' and you will be returned to the CAD screen.

```
Command:
Command:
Knowledge Selection - Examining Existing knowledge...
Enter Knowledge Block Name:
```

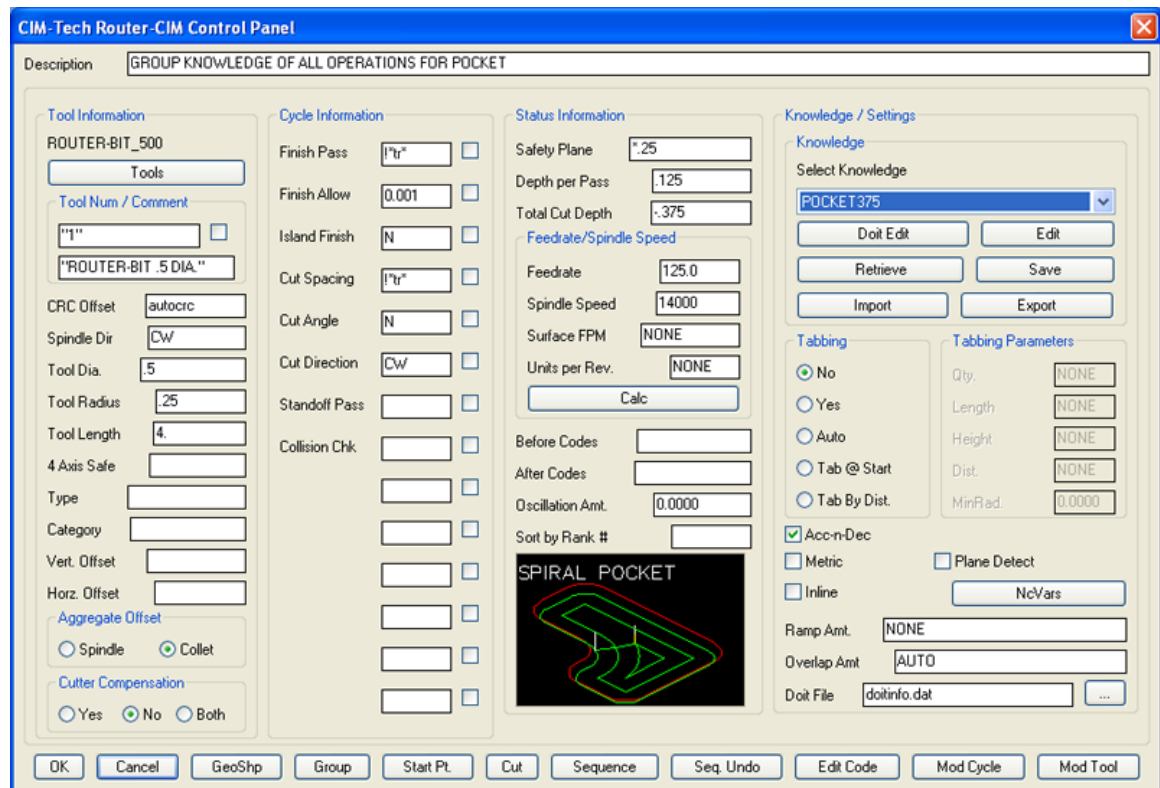
- Here you are prompted to give the Group Knowledge a name.

```
Command:
Command:
Knowledge Selection - Examining Existing knowledge...
Enter Knowledge Block Name: POCKET375
```

- Type in a name (up to 29 characters) and press **ENTER**.

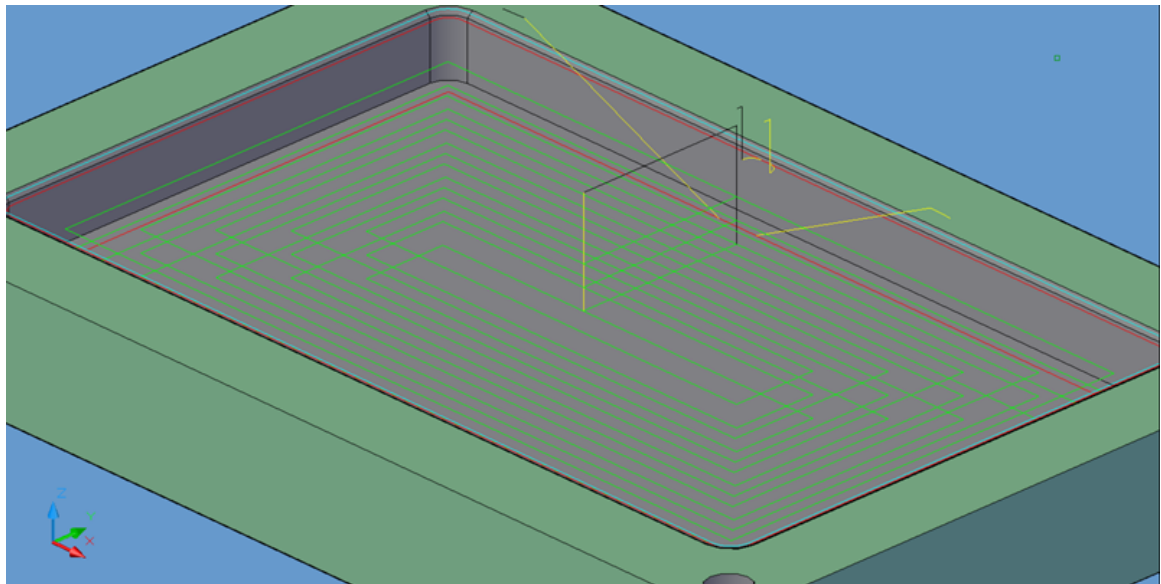
```
Enter Knowledge Block Name: POCKET375
Developing Multiple Knowledge Block...
POCKET375 Knowledge Block Stored.
Command: |
```

- The named Group Knowledge is stored in the drawing in the knowledge list.

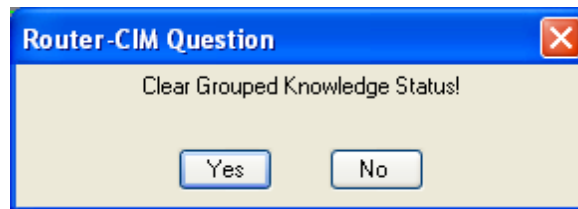


When a Group Knowledge is selected from the list, the first knowledge in the group will appear.

When a shape is selected to cut, all the knowledge operations will be performed in the order in which they are stored in the group. They will be Sequenced in that order as well, unless overridden by another sort key.



Then when you come back into the Control Panel, you will be prompted to clear the group knowledge status.



If you have more cuts to make with this same group right away, you can select '**NO**'. Otherwise, it is advisable to click on '**Yes**', so that you don't accidentally make more than one cut on the next shape.

NOTES: You can **EXPORT** Group Knowledge the same as regular knowledge.

When you use the **Retrieve Knowledge** command to retrieve the Group Knowledge conditions, a '**STATUS**' dialog box will appear for each of the cut conditions contained in the group.

You **CANNOT** use Group Knowledge with **DOIT**, there is no way to determine if you want to clear the Group Status before it goes on to the next knowledge.

Save Without Knowledge

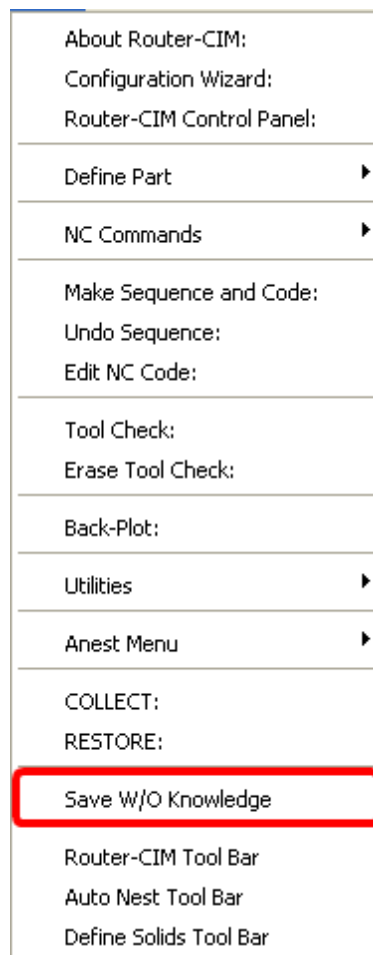


Router-CIM Ribbon:

Keyboard: **SAVEWOKNW**

This command will save the drawing in the smallest file size. It will also remove machine dependent information from the knowledge in the drawing.

This command is found in the RCIM pull down menu.



Save Without Knowledge removes any unused layers, blocks, text styles, and dimension styles, and then saves the drawing. This is the same as the AutoCAD Purge command. This command will purge the drawing three times. By purging three times, any nested blocks are removed.

Your toolpaths will remain, and the knowledge attached to them will remain. However, if a cut is not on the screen, in use, it will be deleted with the rest of the information.

The benefit of this command is that it remove unused, machine dependent information from the drawing. If you are programming several machines and want to use knowledges between them, this helps keep the file size small, while retaining the cut information, and removes the post-processor specific details.

Knowledge Notes

Use of the Knowledge commands will substantially reduce the time required to program a part(s). Once knowledge is created, it may be use repetitively without having to be re-created for each part or drawing.

Each new drawing becomes a knowledge base once you start Router-CIM in it. If the drawing is saved with cuts in it, it is a knowledge base. The \Router-CIM\NCDWGS\"Machine Name\".DWG drawing file is inserted into the current drawing when you load Router-CIM. A knowledge base drawing can contain several hundred blocks. Tools, Cycles, Saved Knowledge, and lead-in/out drawings are among those blocks. Once the part program drawing is complete, the user can strip the extra blocks out of that drawing before saving by using Save Without Knowledge or the AutoCAD Purge command.

Note on Knowledge Base

Users should keep a growing knowledge base on the hard disk. Every time you make a Cut whose cutting condition can be used elsewhere, use the Save Knowledge command and name that condition. This will Save the cutting condition within the drawing. At the end of the drawing session, Export the knowledge(s) to the disk drive.

Important note about Purge

The Purge command will delete any Saved Knowledge that has not been Exported and is not present on the screen when the command is used.

Saved Knowledge is an invisible, not inserted, block that is stored within a drawing. You can see this block listed in the drawing by using the block command and looking for blocks that start with "know_". The Purge command has to be used three times to completely remove data from the drawing. This strips all blocks that are not inserted, so it is important to Export the Saved Knowledge which writes the knowledge(s) to a file on the hard disk.

Tabbing

Tabbing	Tabbing Parameters
<input checked="" type="radio"/> No	Quantity <input type="text" value="NONE"/>
<input type="radio"/> Yes	Length <input type="text" value="NONE"/>
<input type="radio"/> Auto	Height <input type="text" value="NONE"/>
<input type="radio"/> Tab At Start	Distance <input type="text" value="NONE"/>
<input type="radio"/> Tab By Distance	Minimum Radius <input type="text" value="0.00000"/>
	<input type="checkbox"/> Use 3D Tabs
	3D Tab Ramp Amount <input type="text" value="AUTO"/>

Tabbing is a technology that allows smaller or harder to hold parts to stay together by allowing small pieces of material to remain holding the part to the sheet it is being cut from.

There are only a few tabbing settings, and these are easily described in the following section.

Tabbing Settings

Tabbing Settings

Tabbing	Tabbing Parameters
<input checked="" type="radio"/> No	Quantity <input type="text" value="NONE"/>
<input type="radio"/> Yes	Length <input type="text" value="NONE"/>
<input type="radio"/> Auto	Height <input type="text" value="NONE"/>
<input type="radio"/> Tab At Start	Distance <input type="text" value="NONE"/>
<input type="radio"/> Tab By Distance	Minimum Radius <input type="text" value="0.00000"/>
	<input type="checkbox"/> Use 3D Tabs
	3D Tab Ramp Amount <input type="text" value="AUTO"/>

Tabbing

Tabbing will lift the cutter slightly at certain places-selected by you or selected automatically-to leave parts connected to the sheet. Tabbing will allow you to set the Tabs for your part based on the parameters you set in the Control Panel.

New enhanced Tabbing will now allow Tabbing across even the most complex geometry-even when a single Tab may cross multiple elements.

When setting the Length and Height, please refer to this information:

Length: This sets the Length of the Tab, in units. If you leave this field blank, but select either Yes or Automatic to Tabbing, then you will be prompted during the Cut for the correct Tab Length. When setting the Tab Length, determine the desired tab length you need and then ADD the diameter of the tool that you are using in order to get the correct amount of material left of the tab.

NOTE: You can set the length of your tab to a negative number between 0 and 1. Router-CIM will interpret this as a percentage. For example, if you set the Length of the tab to -0.4, you will receive a tab that will be 40% of the total perimeter of the geometry you are tabbing.

Height: This sets the Height of the Tab, in units. If you leave this field blank, but select either Yes, or Automatic to Tabbing, then you will be prompted during the Cut for the correct Tab Height. Determine the height of the tab you need and then you will need to do the calculation since some of the height will be taken off by the tool moving up and over the tab if using 3D tabs.

See example based on using a 1/2 inch tool:

Tab Setting:
Tab Length 1.0
Tab Height 0.25
3D Tab Ramp Amount AUTO

Actual Tab:
Tab Length 0.5
Tab Height 0.125



Tab Setting:
Tab Length 3.0
Tab Height 0.5
3D Tab Ramp Amount AUTO

Actual Tab:
Tab Length 2.5
Tab Height 0.4167



Minimum Radius: This sets the minimum radius that a tab can be placed on. Option is available with Auto and Tab by Distance.

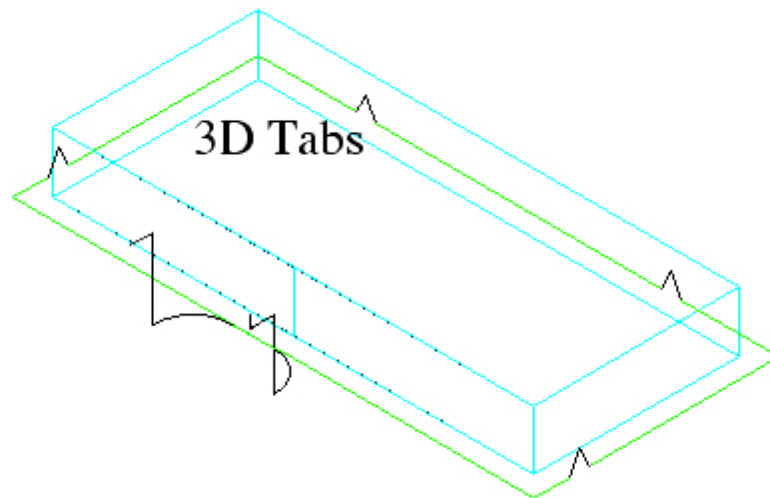
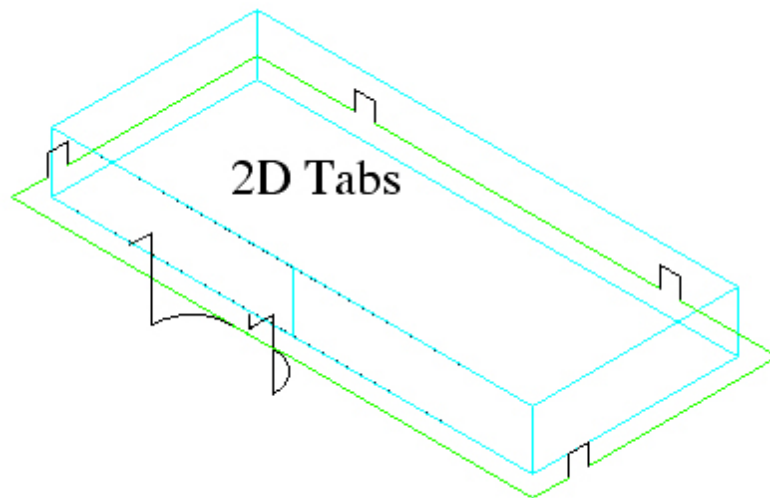
NOTE: When using the Minimum Radius feature with the 'AUTO' tabbing, the cut cycle needs to have the ['Round Corners'](#) option set to N if applicable. The Minimum Radius feature does not support using the Round Corners option.

'Ramped Tabs'

Router-CIM will make 3-D tabs that ramp up and ramp down by default. This is useful for modern machines that use high speed, high precision machining.

Tabbing is either 2D or 3D based on the 'Use 3D Tabs' option:

To learn more about 2D and 3D tabs, please refer to the ['2D or 3D Tabs'](#) section.



See the Tabbing options below:

'NO' to Tabbing

If you select NO, then no Tabbing operations will occur.

'YES' to Tabbing

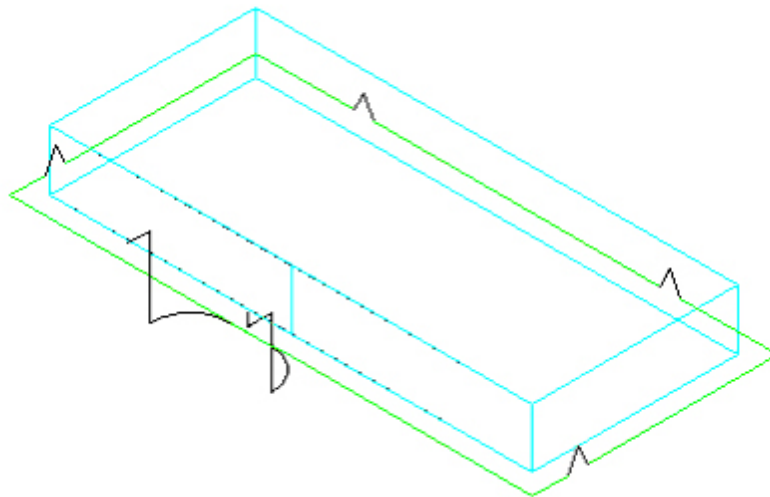
This will set up Tabbing to run and will prompt you for the locations for the Tabs before the Cut is finished. You may pick as many locations as you need for the Tabs.

Note: Be sure to pick on the tool cut path for the Tab location. If you select the geometry for a Tab location, you will not get a Tab placed in that spot.

Tabbing	Tabbing Parameters
<input type="radio"/> No	Quantity <input type="text" value="NONE"/>
<input checked="" type="radio"/> Yes	Length <input type="text" value="1.50000"/>
<input type="radio"/> Auto	Height <input type="text" value="0.25000"/>
<input type="radio"/> Tab At Start	Distance <input type="text" value="NONE"/>
<input type="radio"/> Tab By Distance	Minimum Radius <input type="text" value="0.00000"/>
	<input checked="" type="checkbox"/> Use 3D Tabs
	3D Tab Ramp Amount <input type="text" value="AUTO"/>

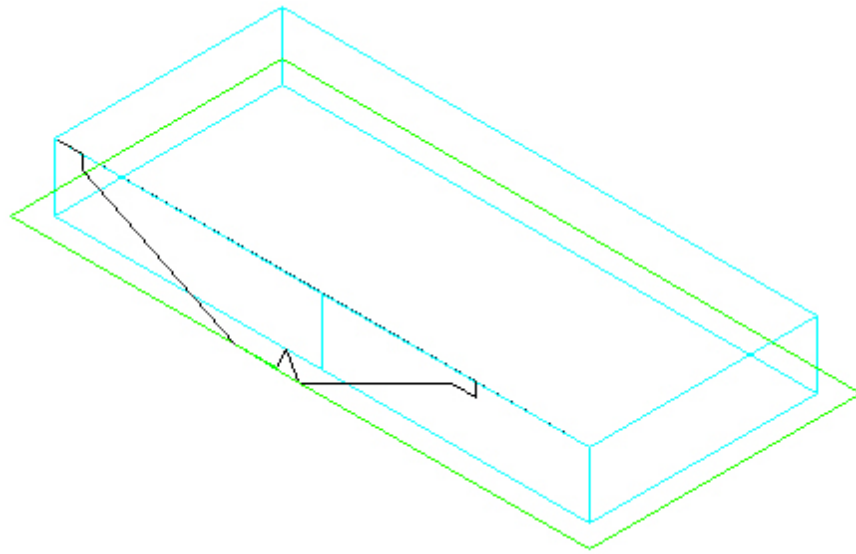
'AUTO' Tabbing

When Automatic is set, you will be prompted for the number of Tabs you wish to have on the part in addition to the Height and Length, and Router-CIM will divide the number of Tabs across the part evenly spaced.

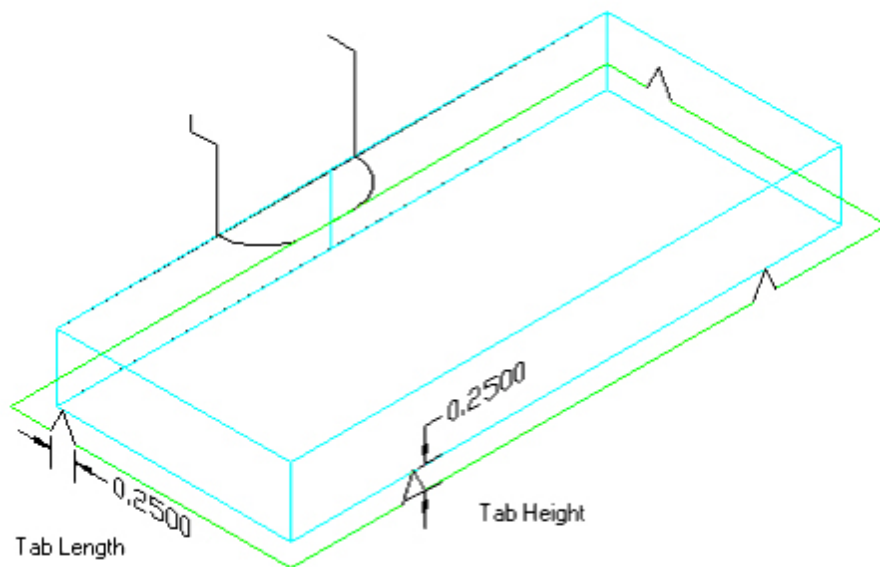


'TAB at Start' Tabbing

When 'TAB at Start' is set, this will produce a Tab immediately after the lead-in. The overlap will automatically cut the Tab off at the end of the cut. This will hold the part at the cut start for the duration of the cut. You will still need to set your Height and Length.

**'TAB by Distance' Tabbing**

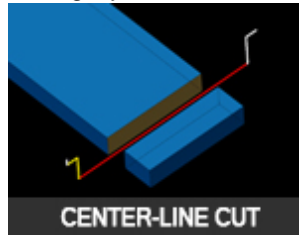
When 'TAB by Distance' is set, you will receive Tabs at the distance interval that you have set. Smaller parts get less tabs while larger parts will receive more tabs. You will still need to set your Height, Length and Distance.

**'Tab Removal'**

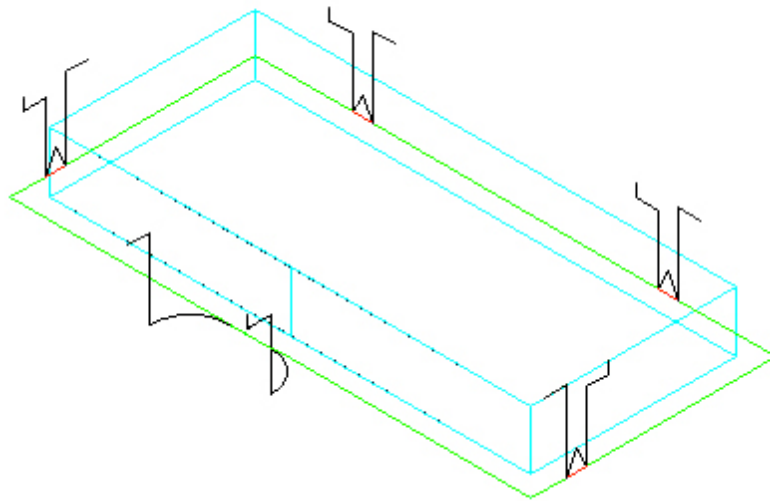
After your parts are initially cut:
Bring up Control Panel/All Stats page

- Set Tabbing from **Yes, Auto or Tab by Distance** to **No**.

- Choose **Center Line Cut** as the Cutting Cycle.



- Click '**OK**' at the bottom left of the Control Panel/All Stats page.
- Type **TABR** at the command line, hit **Enter**. When working within Router-CIM interactive, cut paths will be executed on geometry that is on the TABR layer. It will use the current settings in the Control Panel to accomplish the cuts.

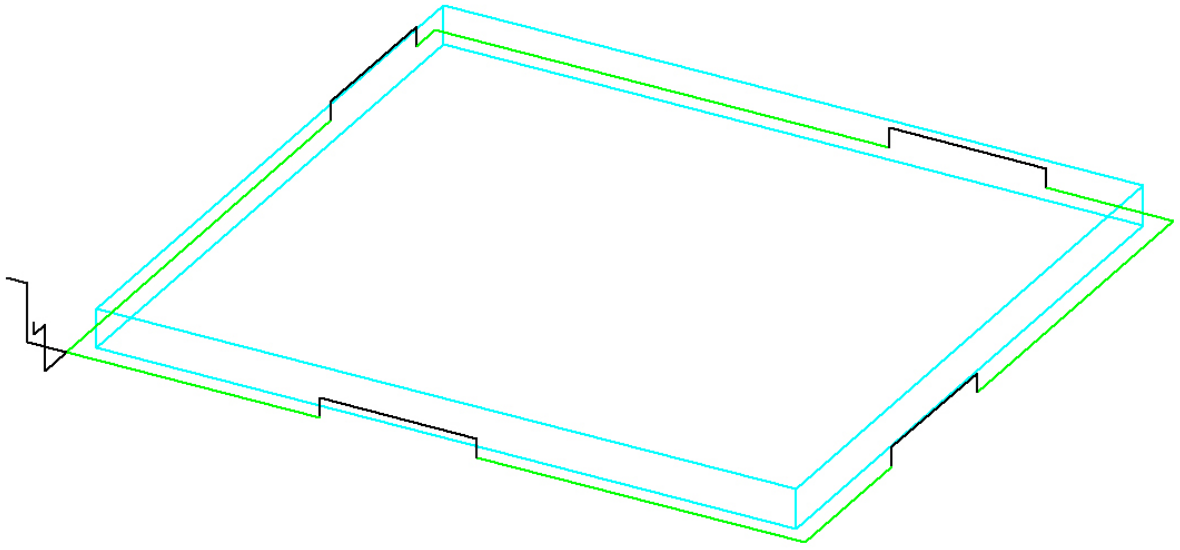


2D or 3D Tabs

There are two methods of tabbing in Router-CIM. You can use either 2D or 3D tabs. The differences are described below.

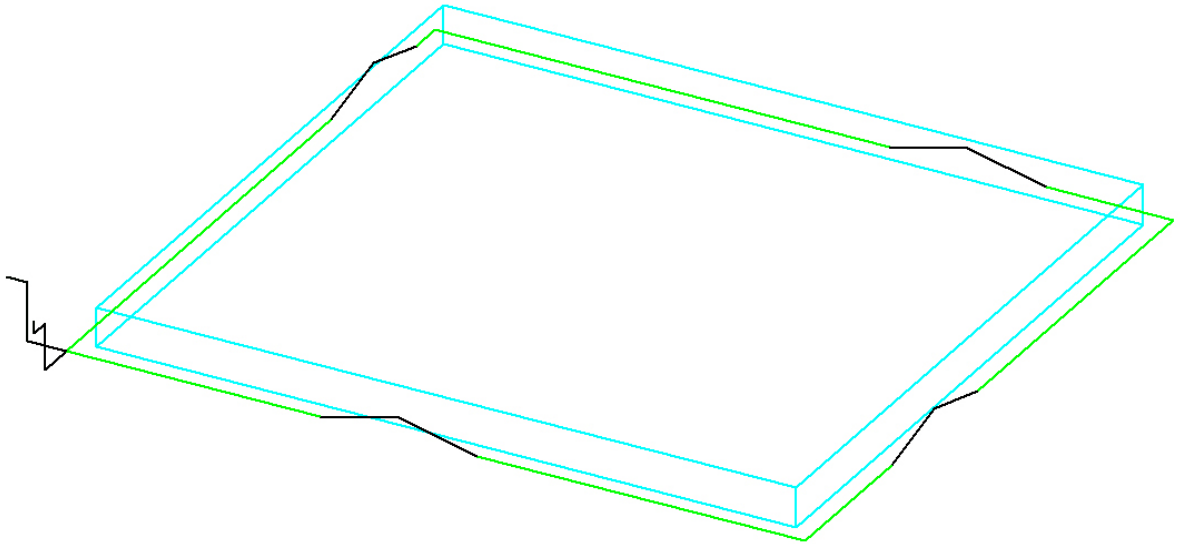
2D Tabs

These tabs are very square, and cause the cutter to stop and lift up in Z to the height of the tab, then move over by the length of the tab and then back down in Z to the normal cut depth.



3D Tabs

3D tabs are usually faster, but leave a varied amount of material. The tool lifts up as it is cutting to the tab height in Z, while the X and/or Y are still moving. Because the tool does not stop and lift up in Z then move over and then move back down in Z in 3 separate moves, the machine will cut these faster.



Changing from 3D Tabs to 2D Tabs

3D tabs are the default in Router-CIM. You can change from 2D tabs to 3D tabs by unchecking the 'Use 3D Tabs' option.

3D Tabs Enabled:

☒ Use 3D Tabs

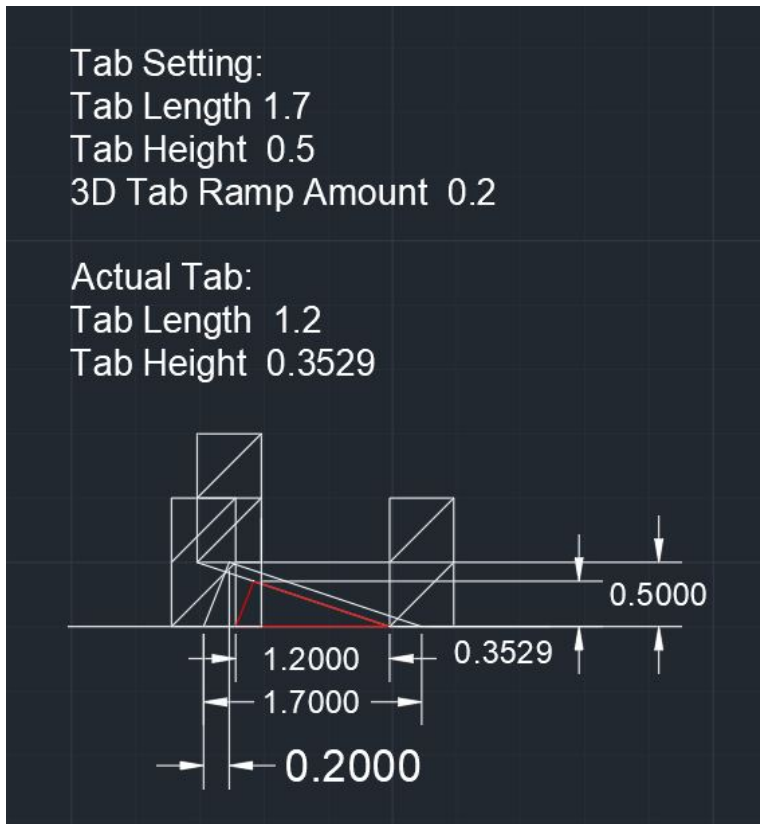
3D Tab Ramp Amount

3D Tabs Disabled:

☐ Use 3D Tabs

3D Tab Ramp Amount

If 3D tabs are enabled, you can adjust the 3D Tab Ramp Amount. The value needs to be numeric to enable the change. The smallest numeric value allowed is 0.01. The default for this field is AUTO



Note: The 3D Tab Ramp Amount only affects the UP movement of the 3D tab. The ramp down movement will be defined by the tab length.

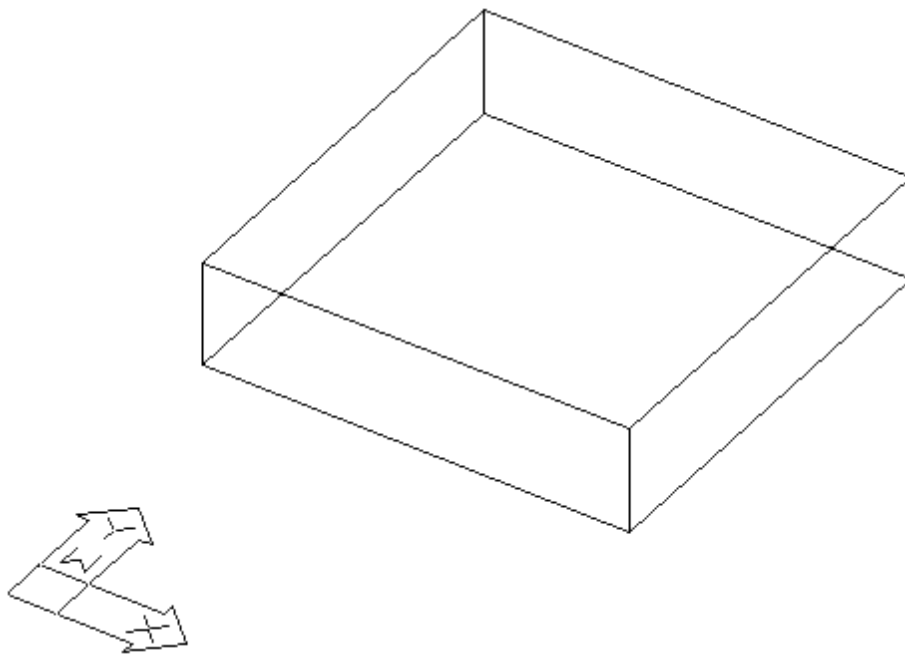
4th Axis Programming

In Router-CIM many types of horizontal routing and drilling may be accomplished with very simple, standard, and familiar knowledge settings. The geometry allowed is the same type of geometry allowed for vertical cuts. For the sake of the machine tool, some care should be taken to ensure that the tool selected is capable of the type of cut specified.

Geometry

Geometry for horizontal applications must be drawn on the face of the part where the shape is desired and should be drawn to finished size. In other words, draw what you want cut, in the finished size. Parts may be drawn with thickness of the desired depth or the depth may be specified during cut. Depending on the type of cut desired the shape can be open or closed. Start points can be modified to suit the desired starting location of the tool.

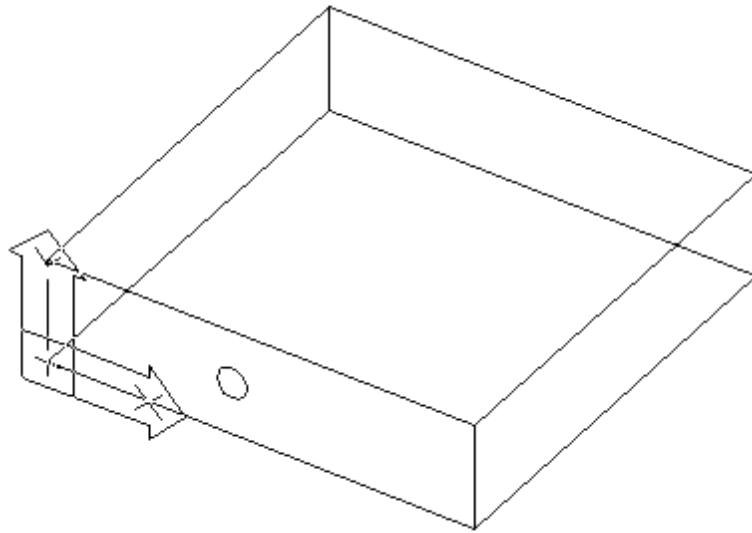
Commands have been provided to allow you to draw geometry on the sides of the part easier. The commands are FS for Front Side, RS for Right Side, BS for Back Side, and LS for Left Side. You may return to the World coordinate system with the UCS, command or typing TS, for Top Side. These are simple UCS switches and you can always use the UCS, 3 point command to create a UCS on any face that does not fall into on one of the 4 sides of a part. If you create a UCS of your own, be sure to use the lower left corner of the part as the origin and that the positive Z direction is away from the part and negative Z is into the part. This way your cut will move into the part and not be inside out. The following example shows two pieces of geometry being created on two different sides of a part.



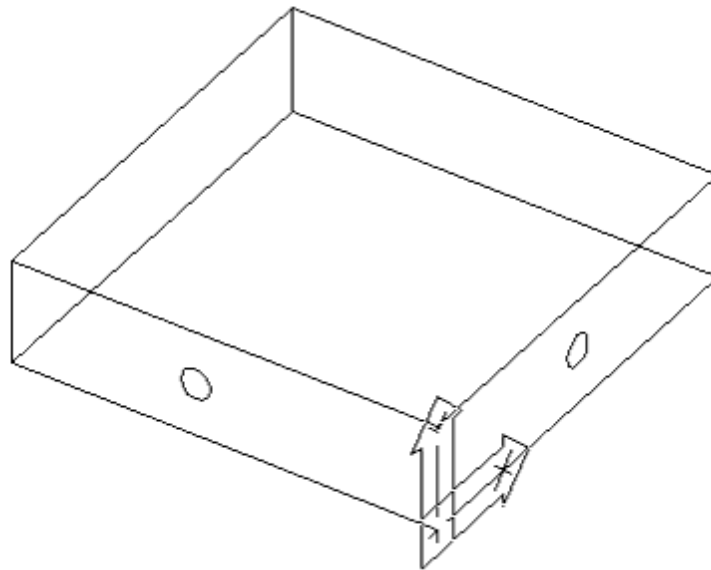
Standard part in World Coordinate System

This part is a standard rectangle drawn with thickness. The UCS Y is pointing at the front side of the part in the illustration above. To draw a shape on the front of your part use the FS command and when prompted for the origin, select the lower left intersection on the front face as your origin.

After the FS command is used, you will have a UCS Icon on the front of your part as shown. You can then draw your shape on the front of your part.



Using the same technique, you can also draw a shape on the right side of the part. Using the RS command, select the lower left corner of the right face. Then draw a circle on the right side. The shapes you have drawn can now be geoshaped and cut using the standard cut cycles. The next section describes the cutting methods necessary.



Cutting

Once the shapes have been geoshaped, you will be able to set up the cut knowledge necessary to cut your shapes. All of the standard cut cycle are available to use for horizontal cutting and only a few considerations are necessary to ensure a proper cut. On the status screen, you should select your cutting tool and cycle. In order to ensure that the horizontal tool will clear the part in the Z axis, a 4 axis

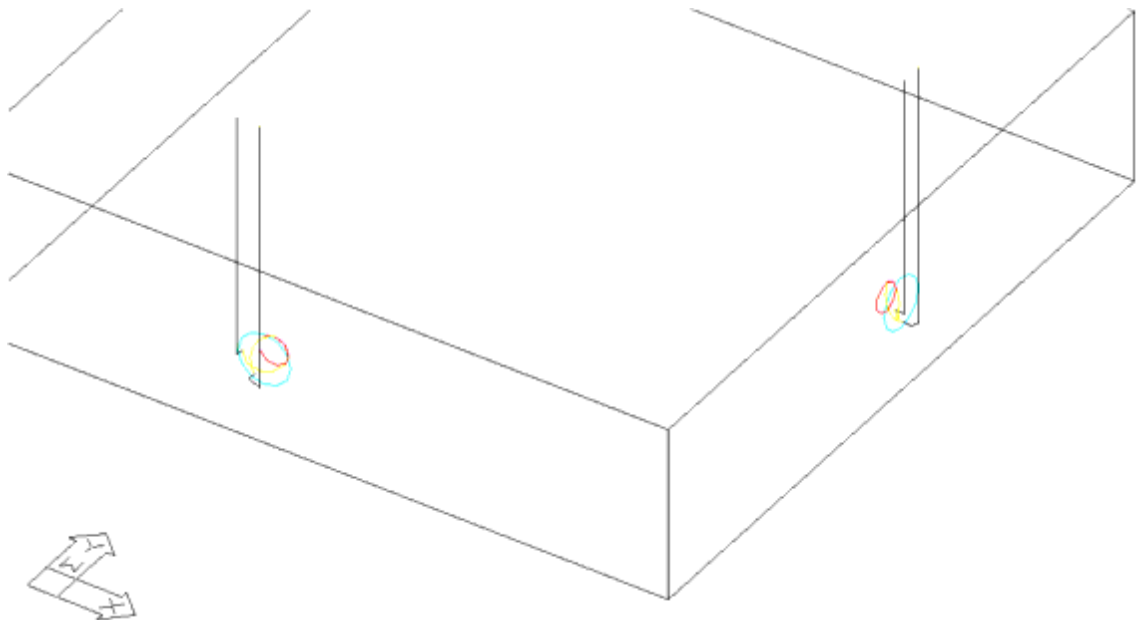
safe height is necessary. The value stored here will be output as the Z clearance height generated before and after the cut. In the lower right side of the screen there is a Plane Detect checkbox that, when activated, will recognize which plane the geometry is drawn on, and generate the correct code for that particular face.

Both of these items, the 4 axis safe and the Plane Detect should be used when making a horizontal cut.

The screenshot displays two panels of settings. The left panel includes: Tab Height (NONE), Acc-n-Dec (unchecked), Metric (unchecked), Plane Detect (checked), Inline (unchecked), Ramp Amt. (NONE), Overlap Amt. (AUTO), and Doit File (doitinfo.dat). The right panel includes: Tool Radius (.1875), Tool Length (4.0), 4 Axis Safe (2.75), Type (empty), Category (empty), Aggregate Offset (Spindle and Collet radio buttons, with Collet selected), and Cutter Compensation (Yes, No, and Both radio buttons, with Both selected).

The 4 Axis Safe and Plane Detect settings on the Status Page.

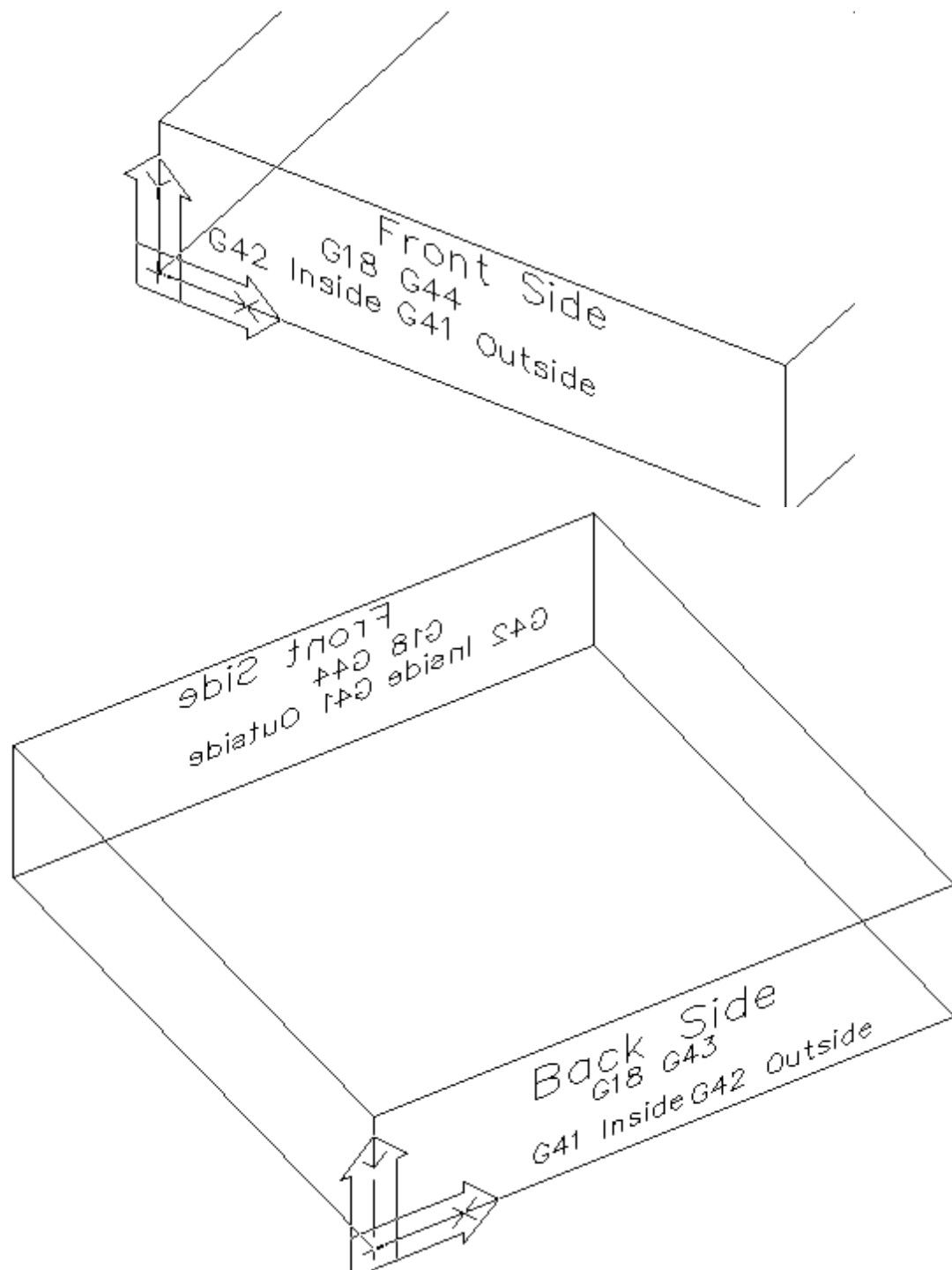
Producing the cuts shown below.



Cutting Planes

When making cuts on the sides of a part, the geometry may lie on one of the planar faces that the machine tool can recognize.

On the Fanuc control, the front side of the part will be the G18 plane and use the G44 command for horizontal length compensation, and cutter compensation will use G42 for inside cuts and G41 for outside cuts.



The Back Side would also be the G18 plane, but will use the G43 command for horizontal length compensation and G41 for cutter compensation on inside cuts and G42 on outside cuts.

The difference in the G43/G44 commands is so the action of the tool as it sets the horizontal length compensation is always the same and POSITIVE values are placed in the offset registers in the controller.

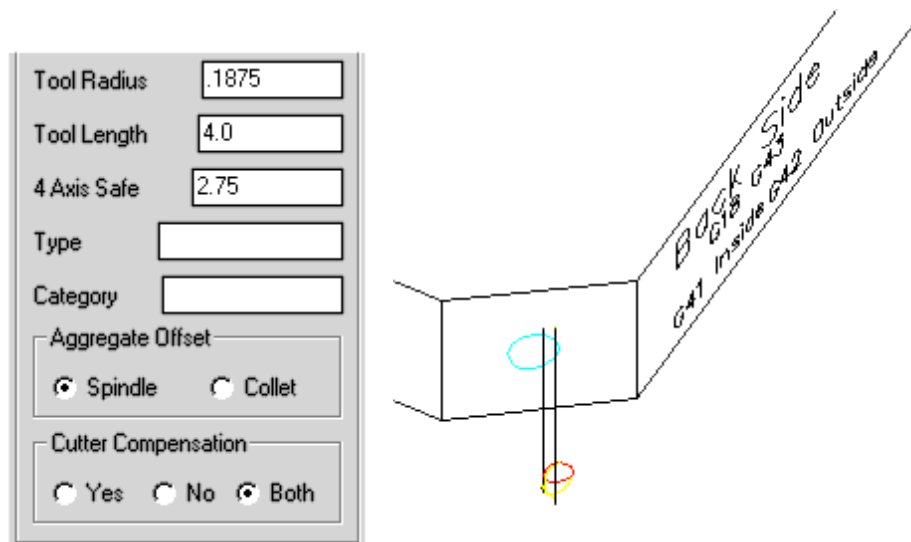
On the left and right sides of the part, the G19 Plane will be called. On the left side, the G19 plane uses the G44 command and G42 inside and G41 on the outside, just like the front face does. The right side of the part uses G19 with G43 and G41 on the inside and G42 on the outside cuts, just like the back side.

In addition to the commands listed some of the G02 and G03 commands for arcs will be reversed as well.

The post processor will generate arcs in the proper direction for the plane selected. If a helical cut is made on a G18 or G19 plane, then G01 point to point moves will be made for the helical portion of the cut and G02/G03 moves made for any flat, planar moves by the cutter.

If the geometry drawn does not lie directly on the G18 or G19 plane, and Plane Detect is checked, then the cuts will be generated as point to point G01 moves in the G17 plane. If this method of cutting is necessary, then another switch is located on the status page that will allow an offset value based on the Tool Length field. The G43/G44 cannot work if the cut is not planar. To accomplish the offset that would normally be done with the G43/G44 command, you must set the toggle Aggregate Offset to Spindle instead of Collet so that the cut is shifted by Tool Length away from the shape.

In the picture shown, the parameters are set as follows:



Setting the Aggregate Offset ensures the offset shown. The Tool Length offset of 4.0 is the distance from the cut to the shape. If the safety plane value is *.25, the depth of cut is -.5, then the tool moves .25 inches away from the shape, then down to the cut position in X and Y, and then lead in from the .25 inch safety plane into the part .5 and then lead out to .25 away from the part and up to Z 2.75 inches (assuming Z0 is top of part). If the Z0 is the top of the table, then the Z value at the start and end will be 2.75 plus the part thickness.

4th Axis Interpolation Cycle

This cycle is to provide continuous four axis motions for aggregate heads and 90° heads on routers.

There is a provision to allow different tool geometry in the toolpaths, such as routers, moulder heads, angled heads.

Typically the geometry provided is still flat 2-D geometry, but 3-D geometry can be provided to all full 4th axis simultaneous programming if the machine allows for this motion.

Drawing Requirements

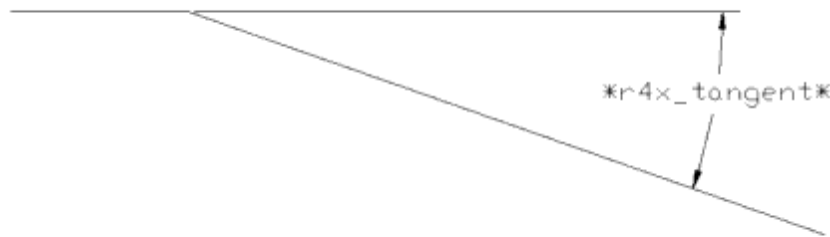
This cycle works on standard geometry created by the Geoshape command and edited with the start point edit command. The shape polylines must be either a two dimensional polyline in the world plane or a three-dimensional shape. Two-dimensional shapes on the side of the part do not need to be interpolated in the fourth axis and any or all of the existing cut cycles will work to create cutting valid cutting motions for these shapes. If an invalid shape is selected, the cycle will tell you when you try to cut it.

The Shape drawn for the Forth Axis Cycle must represent the material location to be cut. The tool tip will extend into the cut shape by the depth specified in the Total Cut Depth specified. The tool path can be shifted down in Z during the cut cycle by specifying the depth in the Move Shape Z field, or the shape can be drawn at the required Z depth. For clarity, we recommend that 3D shapes be drawn at the appropriate Z depth. If a shape has thickness, this will represent the required Z Shift for the Move Shape Z field.

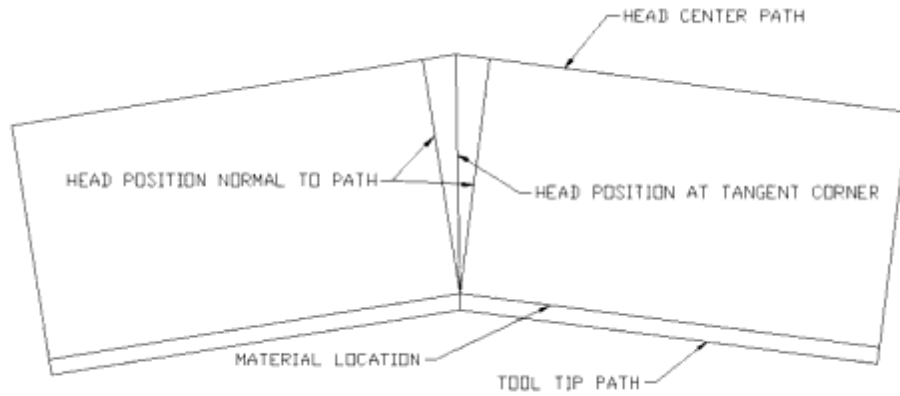
Cycle Corner Treatments

Tangent Corners

The system variable `*r4x_tangent*` defines how far from tangent a corner can diverge and still be called tangent. This variable can be edited with the NCVAR command. The following picture illustrates what is measured to determine tangency.

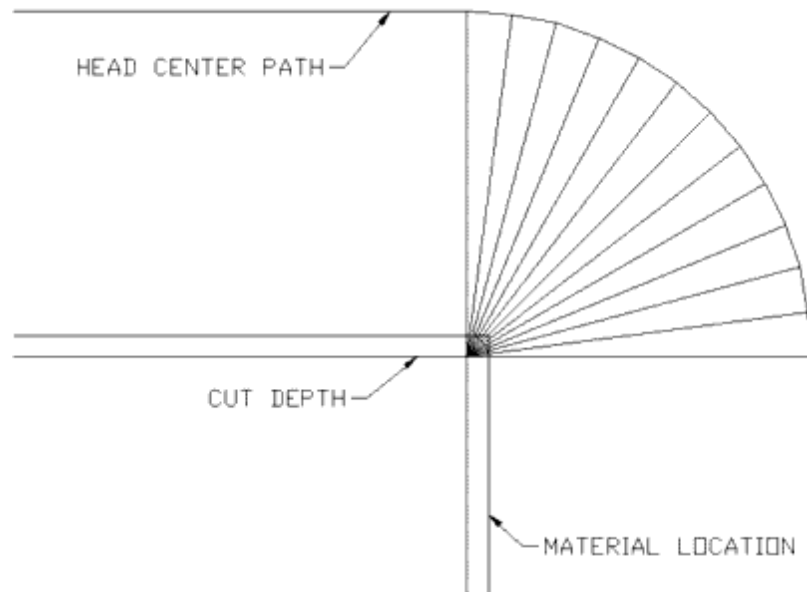


When a tangent corner is found, the motion around the corner smoothly changes the angle of the head to bisect the angles that would represent the position perpendicular to the material. The following illustration shows the head position at the tangent corner as well as the positions that would have been perpendicular to the material to be cut. For clarity, this illustration exaggerates the corner appearance. In normal use, this corner would be much flatter than the illustration.



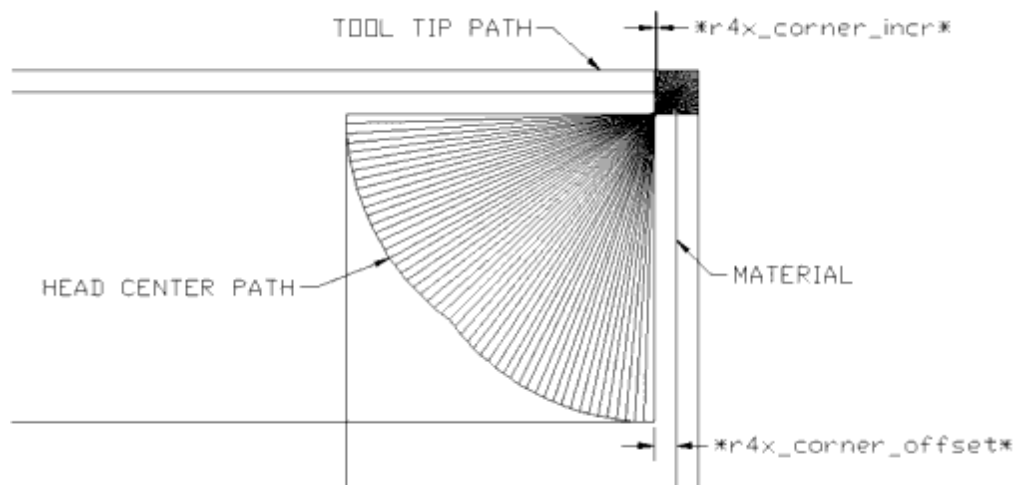
Outside Corners

To get around an outside corner, the corner is cut with a rounded motion. The Head Center Path will swing around the corner while the tool tip remains stationary at the corner point. The following illustration shows this path.



Inside Corners

Inside corners in this application create the motion shown below.



The inside corner logic uses two geometric variables to control the motions in the corner. The two variables are **r4x_corner_offset** and **r4x_corner_incr**. The illustration shows what these two variables control. The head moves perpendicular to the material until it is within **r4x_corner_offset** of the corner to be cut. At that point, the remaining segment to be interpolated is broken by **r4x_corner_incr** and the motions of the head are interpolated around while the angle of the head is changed as shown. When the tool reaches the actual corner, the tool direction will bisect the angle of the corner. Then, the process is reversed until the tool is again **r4x_corner_offset** from the corner and normal to the material.

Position and Direction Control

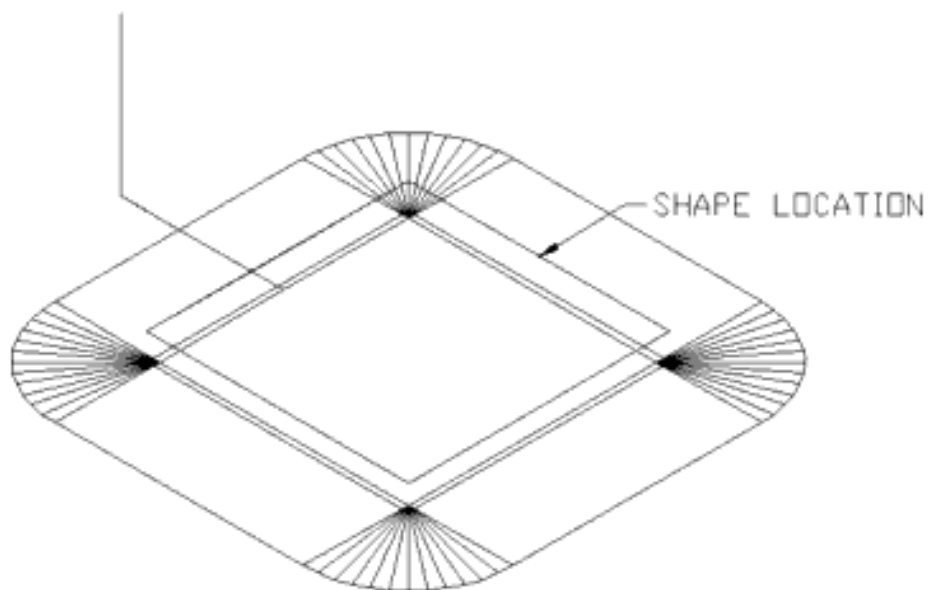
You can change the cut direction from CCW to CW and the cut side from RH to LH. These options work like any other cut cycle in Router-CIM.

Total Cut Depth controls how deep into the material the tip of the tool goes. Depth Per Pass can create multiple passes. The safety plane is measured as the distance from perpendicular to the end of the shape. It is defined in the plane of the leads. If Horizontal leads are specified, the safety moves are horizontal. If Vertical leads are specified, the moves are vertical. Due to concerns on operator and machine safety, ALL moves from the safety plane are made in feed motions. The 4 Axis Safe value must be specified for this cycle, and controls how far above the shape we index.

Move Shape Z is a special parameter for the Fourth Axis Interpolation cycle. It shifts the position of the cut up (positive) or down (negative) in Z by the amount specified. If no amount is specified, and the shape has thickness, the cut will move by this amount.

Cut Side	LH	<input type="checkbox"/>
Cut Direction	CW	<input type="checkbox"/>
Move Shape Z	1	<input type="checkbox"/>

Moves the cut as shown:



Lead Styles

Several styles of leads are supported with the Fourth Axis Interpolation cycle.

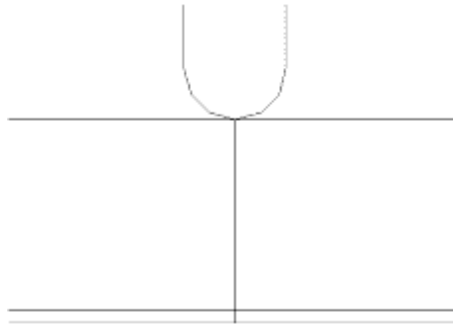
Arc Leads

There are three styles of Arc Leads, depending on the choices made in the Status Page.

Horizontal Arc In

Arc Radius	<input type="text" value="1"/>	<input type="checkbox"/>
Arc Sweep	<input type="text" value="90"/>	<input type="checkbox"/>
Line Length	<input type="text"/>	<input type="checkbox"/>
Line Angle	<input type="text"/>	<input type="checkbox"/>
Vertical Leads	<input type="text"/>	<input type="checkbox"/>
Tool Rotate	<input type="text" value="N"/>	<input type="checkbox"/>

These settings in the Status Page produce the following results:

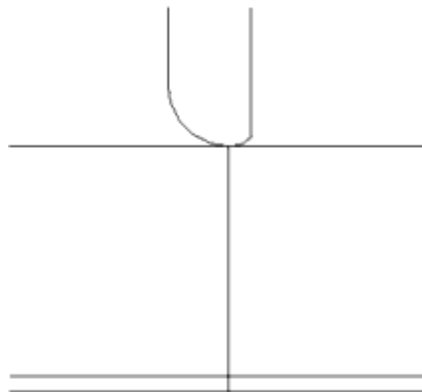


The tool is rotated to normal to the material before the lead in begins, then it moves in to the cut without rotating the tool. The Lead out moves away from the part without rotating the tool.

If a different lead in and lead out is required, two values separated by a space will apply the first value to the lead in and the second value to the lead out. For example, these settings:

Arc Radius	1.5	<input type="checkbox"/>
Arc Sweep	90 45	<input type="checkbox"/>
Line Length		<input type="checkbox"/>
Line Angle		<input type="checkbox"/>
Vertical Leads		<input type="checkbox"/>
Tool Rotate	N	<input type="checkbox"/>

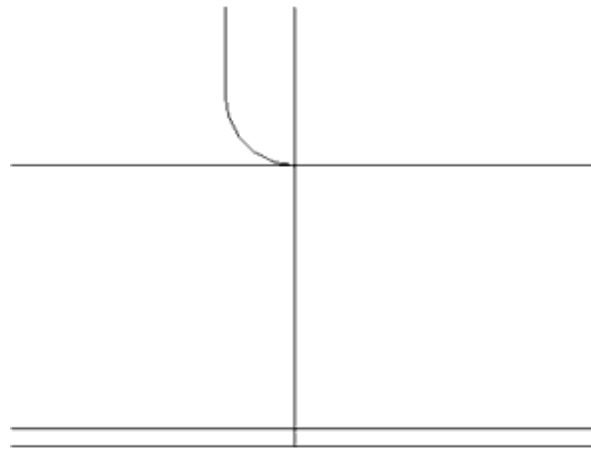
Produces the following result:



The lead in is the same; the lead out has changed. If you wish to suppress one or the other, put an **N** in place of a value as shown:

Arc Radius	1 N	<input type="checkbox"/>
Arc Sweep	90 N	<input type="checkbox"/>
Line Length		<input type="checkbox"/>
Line Angle		<input type="checkbox"/>
Vertical Leads		<input type="checkbox"/>
Tool Rotate	N	<input type="checkbox"/>

The **N** in the second position will suppress the lead out:



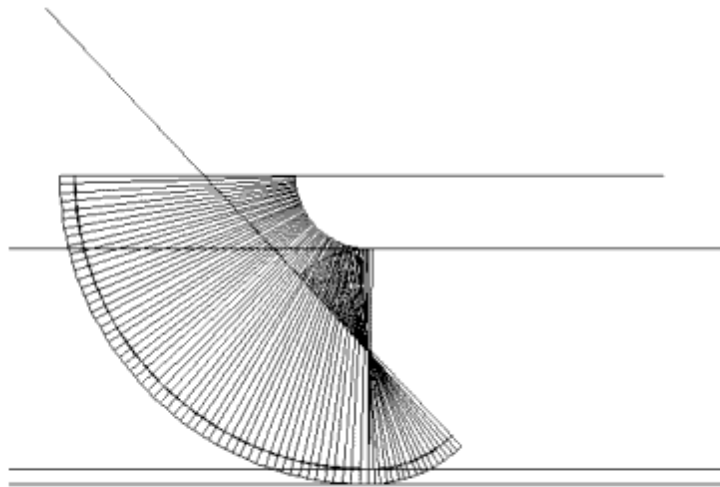
An arc lead in can only be created when both the radius and the sweep have been defined.

Horizontal Arc Leads with Tool Rotate

When you turn on tool rotate for the arc leads, the tool will sweep in on the arc. The following settings illustrate some of the actions that can be performed with the tool rotate option.

Arc Radius	5 2	<input type="checkbox"/>
Arc Sweep	90 45	<input type="checkbox"/>
Line Length		<input type="checkbox"/>
Line Angle		<input type="checkbox"/>
Vertical Leads		<input type="checkbox"/>
Tool Rotate	<input checked="" type="checkbox"/> Y	<input type="checkbox"/>

Produces the following results:



The tool is rotated 90 degrees to the shape, moves into the arc and sweeps by rotating the head as the tool center moves on the arc. In this example, the head is defined as 4 units long. On the lead out, the arc is smaller than the tool radius, so the reverse arc effect is shown as the tool pivots around 45 degrees before backing away from the part.

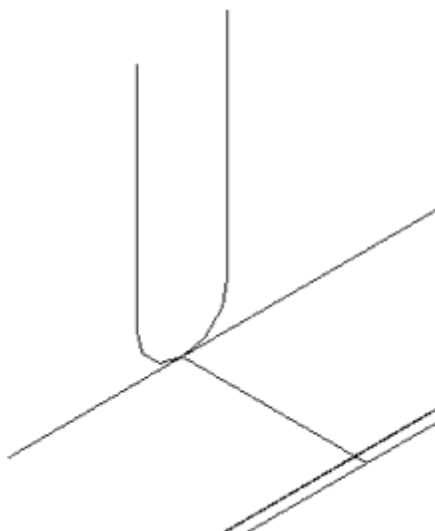
The tool rotate option can also be used selectively on the lead in and out by separating the values for the lead in and lead out by spaces. Tool Rotate only works on Horizontal Arc Leads.

Vertical Arc Leads

When the vertical option is turned on, the arc is created vertically instead of in the XY Plane.

Arc Radius	1	<input type="checkbox"/>
Arc Sweep	90	<input type="checkbox"/>
Line Length		<input type="checkbox"/>
Line Angle		<input type="checkbox"/>
Vertical Leads	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Tool Rotate	N	<input type="checkbox"/>

The results of setting vertical leads is shown below in the isometric view:



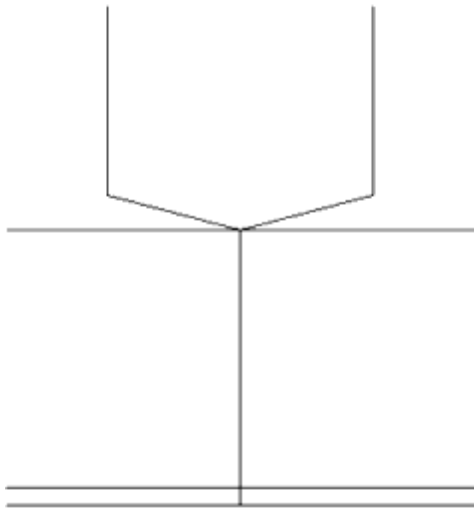
The tool is rotated perpendicular to the shape, then brought down onto the shape. As always, you can have different behaviors of the lead in and out by separating the values by spaces.

Horizontal Line Leads

The length of the line and the angle away from tangent required defines Line Leads. For Example:

Arc Radius	<input type="text"/>	<input type="checkbox"/>
Arc Sweep	<input type="text"/>	<input type="checkbox"/>
Line Length	<input type="text" value="2"/>	<input type="checkbox"/>
Line Angle	<input type="text" value="15"/>	<input type="checkbox"/>
Vertical Leads	<input type="text"/>	<input type="checkbox"/>
Tool Rotate	<input type="text" value="N"/>	<input type="checkbox"/>

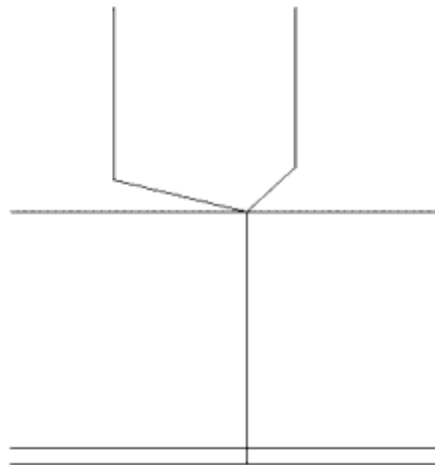
Produces the following results:



Like the arc leads, different values can be entered to achieve different results for the lead in verses the lead out.

Arc Radius	<input type="text"/>	<input type="checkbox"/>
Arc Sweep	<input type="text"/>	<input type="checkbox"/>
Line Length	<input type="text" value="21"/>	<input type="checkbox"/>
Line Angle	<input type="text" value="15.45"/>	<input type="checkbox"/>
Vertical Leads	<input type="text"/>	<input type="checkbox"/>
Tool Rotate	<input type="text" value="N"/>	<input type="checkbox"/>

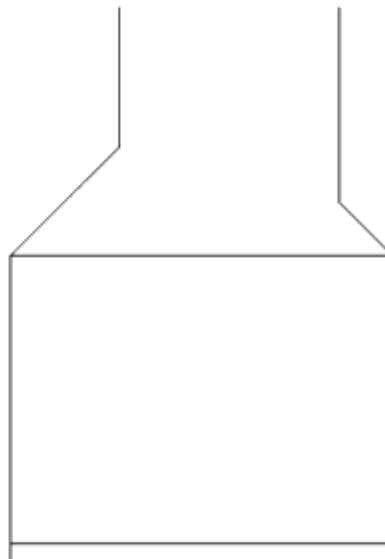
Produces



If the angle entered is greater than 90 degrees, the line will cut backwards to the lead point. This can be useful for entering slot cuts on the side of the part with a ramp in motion:

Arc Radius	<input type="text"/>	<input type="checkbox"/>
Arc Sweep	<input type="text"/>	<input type="checkbox"/>
Line Length	<input type="text" value="21"/>	<input type="checkbox"/>
Line Angle	<input type="text" value="135"/>	<input type="checkbox"/>
Vertical Leads	<input type="text"/>	<input type="checkbox"/>
Tool Rotate	<input type="text" value="N"/>	<input type="checkbox"/>

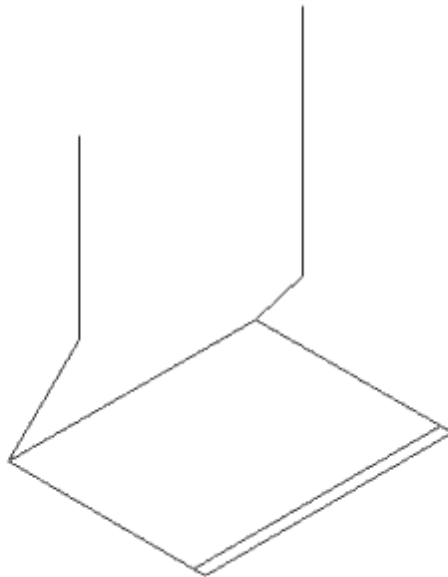
When these settings are cut on a simple line shape representing a slot cut, you get the following results:



Vertical Line Leads

Line leads can be vertical as well:

Arc Radius	<input type="text"/>	<input type="checkbox"/>
Arc Sweep	<input type="text"/>	<input type="checkbox"/>
Line Length	<input type="text" value="21"/>	<input type="checkbox"/>
Line Angle	<input type="text" value="135 15"/>	<input type="checkbox"/>
Vertical Leads	<input type="text" value="Y"/>	<input type="checkbox"/>
Tool Rotate	<input type="text" value="N"/>	<input type="checkbox"/>



The lead in is a backward facing move, the lead out tapers off the end of the cut. Mixing vertical with horizontal only requires Y or N separated by a space as required.

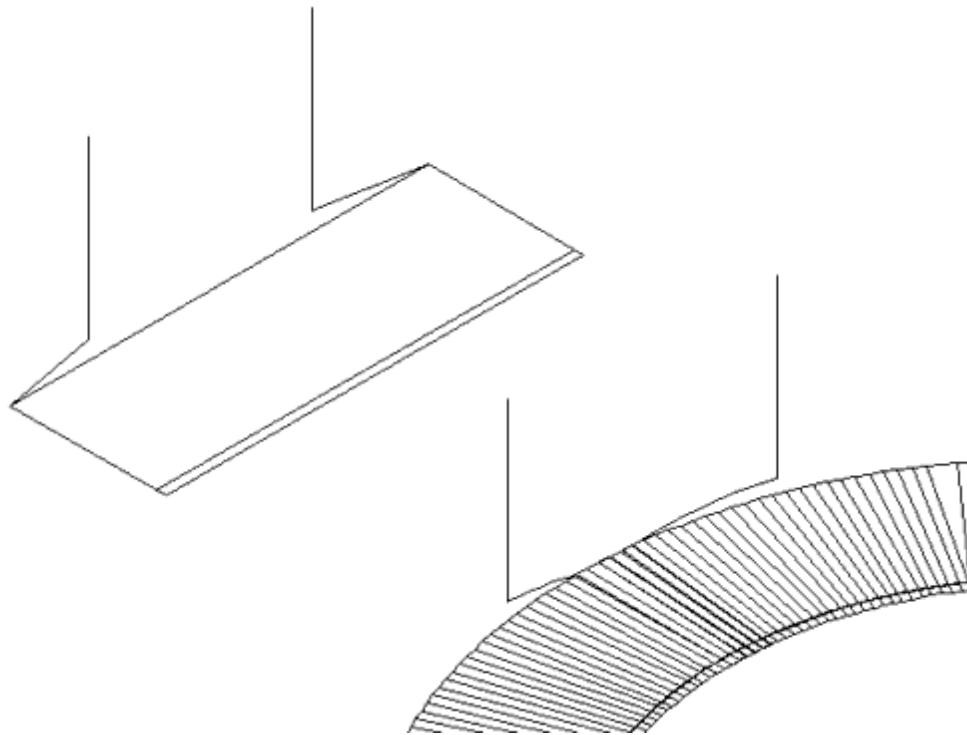
Channel Leads

Channel lead in and out is available as well. They are always vertical, and do not appear when cutting 3D geometry. When the geometry is open, the lead in will cut back along the contour until it gets to the lead in point, then it will cut forward along the shape. The channel out will cut to the end of the shape then back up along the shape up and out.

Overlap is often used with channel style leads, but it can be used with the other lead in styles as well. The Overlap parameter only works on closed shape. On open shapes you do not get an overlap.

Arc Radius	<input type="text"/>	<input type="checkbox"/>
Arc Sweep	<input type="text"/>	<input type="checkbox"/>
Line Length	<input type="text"/>	<input type="checkbox"/>
Line Angle	<input type="text"/>	<input type="checkbox"/>
Vertical Leads	<input type="text"/>	<input type="checkbox"/>
Tool Rotate	<input type="text" value="N"/>	<input type="checkbox"/>
Overlap	<input type="text" value="1"/>	<input type="checkbox"/>
Channel Dist	<input type="text" value="23"/>	<input type="checkbox"/>

When cut on an open and closed shape produces these results:



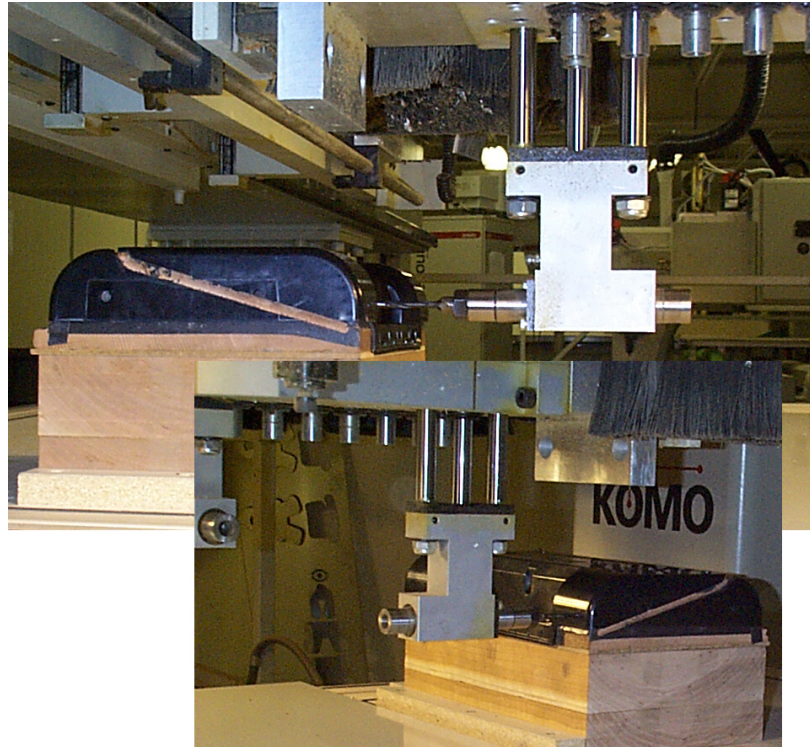
Notice that the leads follow the shape of the cut as they taper down into the shape. The overlap only occurred on the closed shape. The direction of the lead in and lead out reversed on the open shape. Familiarity with the commands for your particular machine will ensure that you can recognize the code as generated by the post processor.

The use of specific aggregate tooling may require slight modification to these parameters, and CIM-Tech is available to assist you in configuring these parameters to suit your needs.

Horizontal Boring

Many machine tools have horizontal boring capabilities. Some machines have boring blocks with horizontal drills, some have aggregate tools with horizontal boring capability, and some have both.

The following section is a short tutorial covering some methods that you can use in Router-CIM to make horizontal boring tool paths. There is also some discussion about setting up offsets and work coordinates depending on the methods of cutting you use and how your machine tool is equipped.



Horizontal Boring

There are 4 possible ways to do horizontal boring operations in Router-CIM with a Boring Block. There is a drawing attached to this tutorial as a sample and there is a knowledge for each one of the 4 methods. This document will explain how they would work on the machine.

The drawing has holes on each side, and for horizontal boring you must draw the circle on the correct Face of the part.

To help you with this, Router-CIM has 4 UCS commands to help fix the UCS on the correct Face.

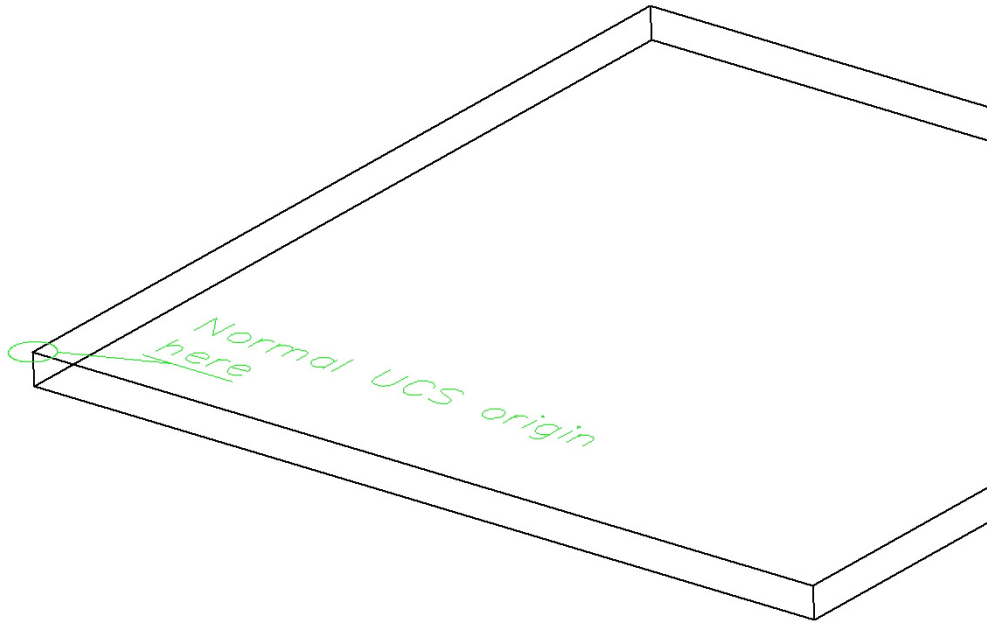
The FS command will ask for the origin point (lower left corner) of the Front side of the part.
The RS command will ask for the origin point (lower left corner) of the Right side of the part.
The BS command will ask for the origin point (lower left corner) of the Back side of the part.
The LS command will ask for the origin point (lower left corner) of the Left side of the part.

If the parts are not drawn on the correct face, the machine will not cut them properly.

Using FS, RS, BS, LS

To use the Router-CIM quick UCS commands, you simply need to enter the command (FS, RS, BS, LS) at the command prompt, and when prompted for the Origin location, select the Lower Left Corner of the part.

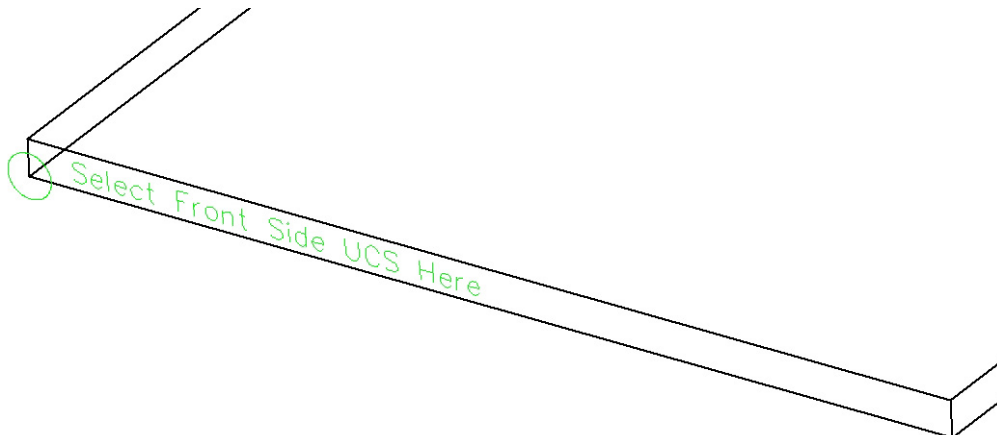
The normal World Coordinate system is set to the lower left corner of the top of the part:



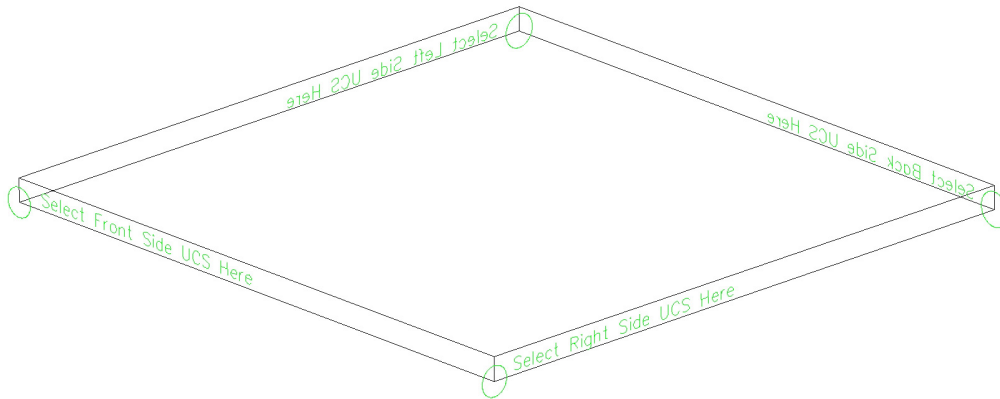
To use the side UCS commands, you need to select the lower left corner of the face you are going to work on. For example, when using FS, you will be prompted:

`fs Origin point <0,0,0>:`

From this prompt, select the Lower Left corner of the front side of the part



To demonstrate, there is a drawing showing the correct point of each of the 4 sides.



The basic point to make is that after entering any of the 4 quick UCS commands, you must choose the Lower Left Corner of the face you are about to work on.

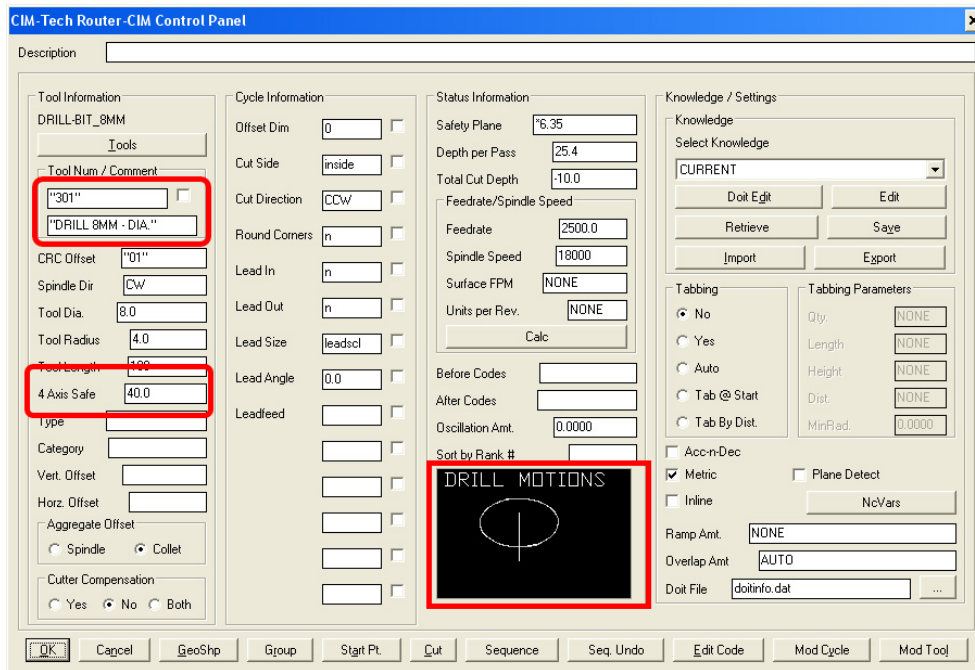
Setting up Knowledge

Setting up Knowledge for Horizontal Boring

As mentioned earlier, there are 4 ways to drill horizontally, each being slightly different both at the machine and in Router-CIM.

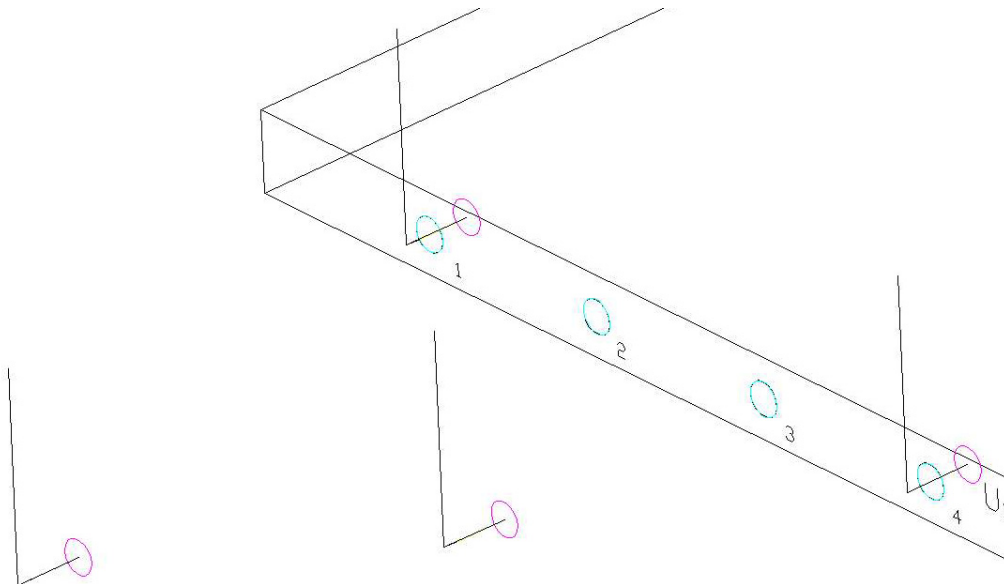
Each of the types of horizontal boring use the Drill Motions cycle, and you must select the correct Tool Number for the face you are going to cut.

In addition, each uses the 4 Axis Safe position to move the drill up above the part between cuts.



Included in the Sample Drawing are 4 knowledges for each face. These represent the 4 different methods of cutting horizontally with the boring block and also use the correct tool numbers for each face.

This document will use the Front Side Drills as an example.



FrontSide1

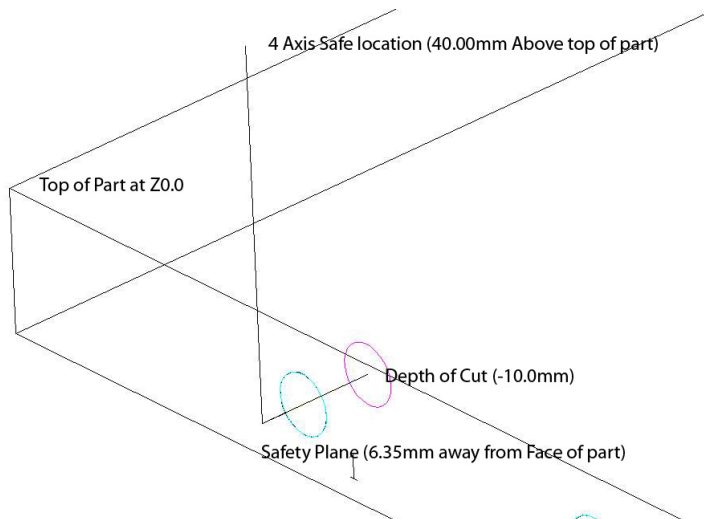
The FrontSide1 knowledge is a very basic boring knowledge and is the easiest to set up in Router-CIM, but not the easiest to set up at the machine.

The screenshot shows the Router-CIM software interface with the following settings:

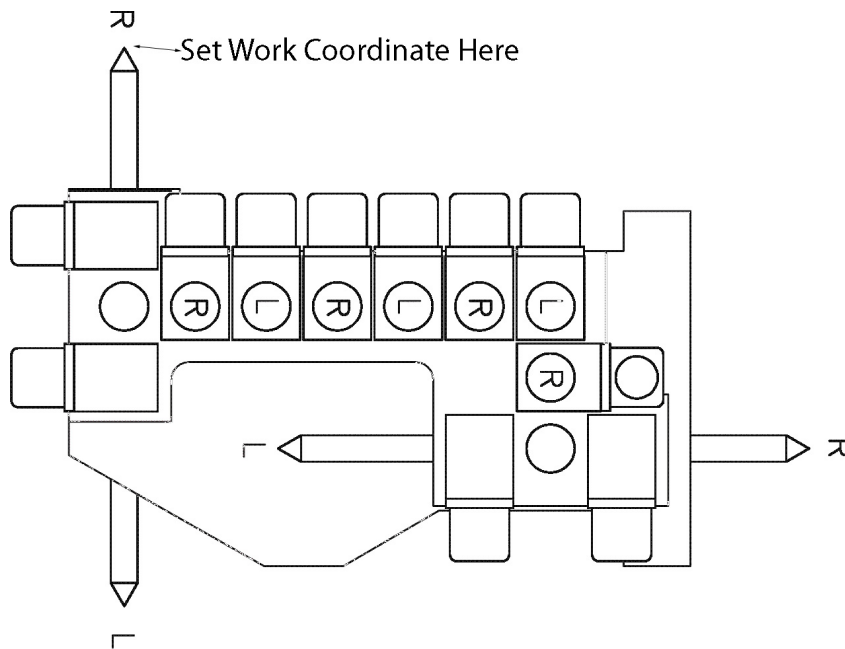
- Tool Information:**
 - DRILL-BIT_8MM
 - Tools: [Tools]
 - Tool Num / Comment: "301" and "DRILL 8MM - DIA."
 - CRC Offset: "01"
 - Spindle Dir: CW
 - Tool Dia: 8.0
 - Tool Radius: 4.0
 - Tool Length: 100
 - 4 Axis Safe: 40.0
 - Type: [Blank]
 - Category: [Blank]
 - Vert. Offset: [Blank]
 - Horz. Offset: [Blank]
 - Aggregate Offset: Spindle (selected), Collet (selected)
 - Cutter Compensation: Yes (selected), No (selected), Both (selected)
- Cycle Information:**
 - Offset Dim: 0
 - Cut Side: inside
 - Cut Direction: CCW
 - Round Corners: n
 - Lead In: n
 - Lead Out: n
 - Lead Size: leadscl
 - Lead Angle: 0.0
 - Leadfeed: [Blank]
- Status Information:**
 - Safety Plane: 6.35
 - Depth per Pass: 25.4
 - Total Cut Depth: -10.0
 - Feedrate/Spindle Speed:
 - Feedrate: 2500.0
 - Spindle Speed: 18000
 - Surface FPM: NONE
 - Units per Rev: NONE
 - Before Codes: [Blank]
 - After Codes: [Blank]
 - Oscillation Amt: 0.0000
 - Sort by Rank #: [Blank]
- Knowledge / Settings:**
 - Knowledge: FRONTSIDE1
 - Doit Edit: [Blank]
 - Edit: [Blank]
 - Retrieve: [Blank]
 - Save: [Blank]
 - Import: [Blank]
 - Export: [Blank]
 - Tabbing:
 - No (selected)
 - Yes
 - Auto
 - Tab @ Start
 - Tab By Dist.
 - Tabbing Parameters:
 - Qty: NONE
 - Length: NONE
 - Height: NONE
 - Dist: NONE
 - MinRad: 0.0000
 - Acc-n-Dec: [Blank]
 - Metric: [Blank]
 - Inline: [Blank]
 - Plane Detect: [Blank]
 - Ramp Amt: NONE
 - Overlap Amt: AUTO
 - Doit File: doitinfo.dat

This knowledge uses a 4 Axis safe and has the Aggregate Offset set to Collet and Plane Detect is Not selected.

When this knowledge performs a cut, the tool path shows up in a normal location with the cut inside the circle. The Collet setting keeps the cut in this location. The 4 Axis Safe setting makes sure the tool moves to 40.0mm above the top of the part both before and after the cut. The Safety Plane is now a horizontal move to 6.35mm away from the front face of the part.



The important idea with this knowledge is that for the cut to be in the correct position on the part, the Tip of the drill bit must be set into the Work Coordinate at the correct X, Y location or else this tool cannot reach the correct point of the cut. When you touch off the tip of the vertical drills to the corner of the table, then the horizontal drill tips must be set there too.

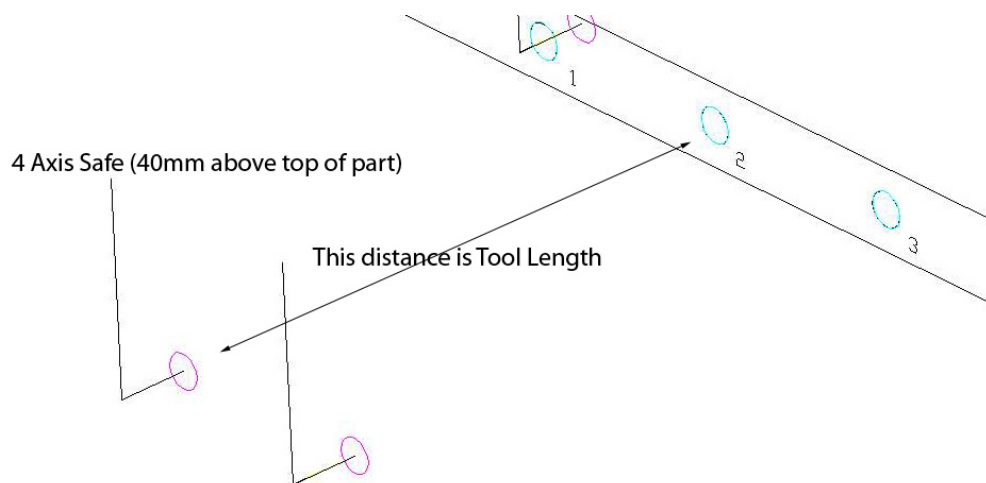


FrontSide2

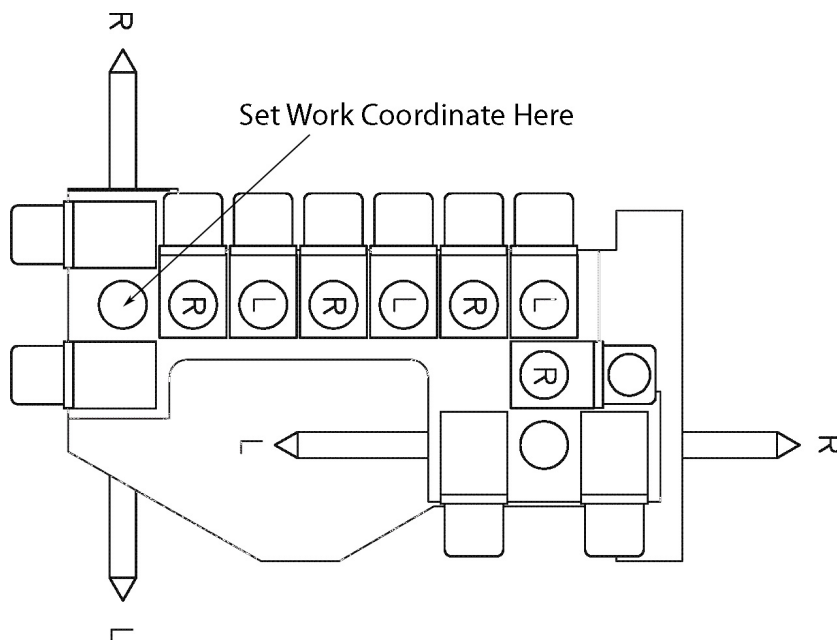
The FrontSide2 knowledge looks a little strange on the screen (it is away from the part) but is actually a little easier to use at the machine. The settings for this knowledge are:

Tool Information	Cycle Information	Status Information	Knowledge / Settings
DRILL-BIT_8MM Tools Tool Num / Comment "301" "DRILL 8MM - DIA." CRC Offset "01" Spindle Dir CW Tool Dia. 8.0 Tool Radius 4.0 Tool Length 150.0 4 Axis Safe 40.0 Type Category Vert. Offset Horiz. Offset Aggregate Offset <input checked="" type="radio"/> Spindle <input type="radio"/> Collet Cutter Compensation <input type="radio"/> Yes <input checked="" type="radio"/> No <input type="radio"/> Both	Offset Dim 0 Cut Side inside Cut Direction CCW Round Corners n Lead In n Lead Out n Lead Size leadscl Lead Angle 0.0 Leadfeed	Safety Plane 6.35 Depth per Pass 25.4 Total Cut Depth -10.0 Feedrate/Spindle Speed Feedrate 2500.0 Spindle Speed 18000 Surface FPM NONE Units per Rev. NONE Calc Before Codes After Codes Oscillation Amt. 0.0000 Sort by Rank # DRILL MOTIONS	Knowledge Select Knowledge FRONTSIDE2 Doit Edit Edit Retrieve Save Import Export Tabbing <input checked="" type="radio"/> No <input type="radio"/> Yes <input type="radio"/> Auto <input type="radio"/> Tab @ Start <input type="radio"/> Tab By Dist Tabbing Parameters Qty. NONE Length NONE Height NONE Dist. NONE MinRad. 0.0000 <input checked="" type="checkbox"/> Acc-n-Dec <input checked="" type="checkbox"/> Metric <input type="checkbox"/> Plane Detect <input type="checkbox"/> Inline Ramp Amt. NONE Overlap Amt. AUTO Doit File doitinfo.dat

For this knowledge, you use the Tool Length to set the X, Y, offset from the center of the drill spindle and use the Spindle setting in Aggregate Offset. Plane Detect is not used for this method either.

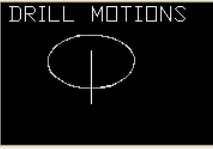


In order for this to work properly, you must know the distance from the tip of the drill to the center of the drill block location (where the drill would be if it were a vertical drill). This is where the Work Coordinate would normally be set to. Then, put that distance into the Tool Length location. Router-CIM will move the tool path away from the part by that amount so that the TIP of the tool is actually at the right location. This requires no input on the machine operators side other than possibly informing the programmer of the length of the drill from the work coordinate location.

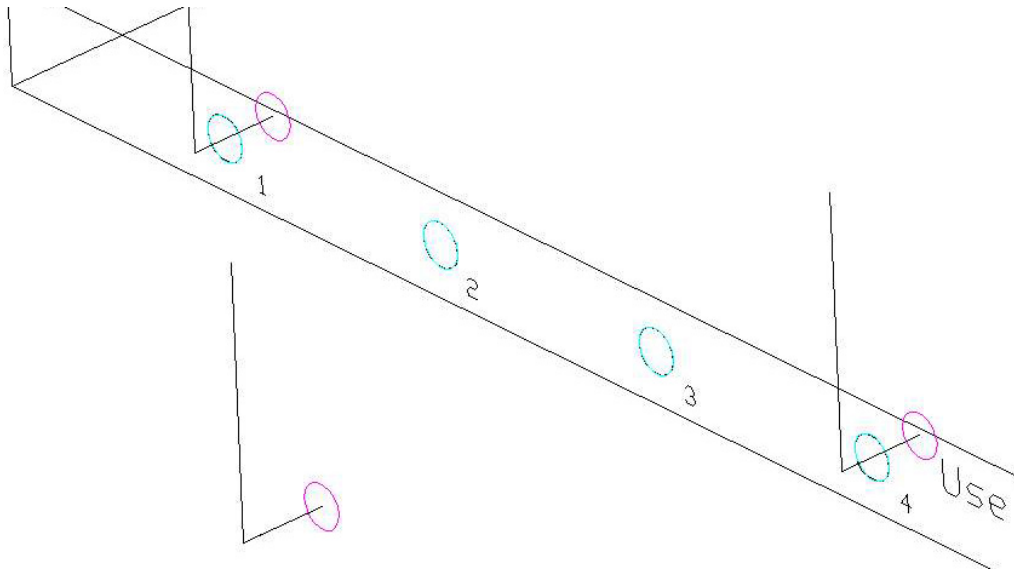


FrontSide3

This method is slightly different from the other two in that it will use Plane Detect and the code will be slightly different. There will also be a setting in the code for a horizontal offset to be input in the machine.

Tool Information DRILL-BIT_8MM Tools Tool Num / Comment "301" "DRILL 8MM - DIA." CRC Offset "01" Spindle Dir CW Tool Dia 8.0 Tool Radius 4.0 Tool Length 80.0 4 Axis Safe 40.0 Type Category Vert. Offset Horiz. Offset Aggregate Offset Spindle Collet Cutter Compensation Yes No Both	Cycle Information Offset Dim 0 Cut Side inside Cut Direction CCW Round Corners n Lead In n Lead Out n Lead Size leadscl Lead Angle 0.0 Leadfeed	Status Information Safety Plane "6.35" Depth per Pass 25.4 Total Cut Depth -10.0 Feeds/Spindle Speed Feedsrate 2500.0 Spindle Speed 18000 Surface FPM NONE Units per Rev. NONE Calc Before Codes After Codes Oscillation Amt. 0.0000 Sort by Rank # DRILL MOTIONS 	Knowledge / Settings Knowledge Select Knowledge FRONTSIDE3 Doit Edit Edit Retrieve Save Import Export Tabbing No Yes Auto Tab @ Start Tab By Dist Tabbing Parameters Qty. NONE Length NONE Height NONE Dist. NONE MinRad. 0.0000 Acc-n-Dec Metric Plane Detect Inline Ncvars Ramp Amt. NONE Overlap Amt. AUTO Doit File doitinfo.dat
---	---	---	---

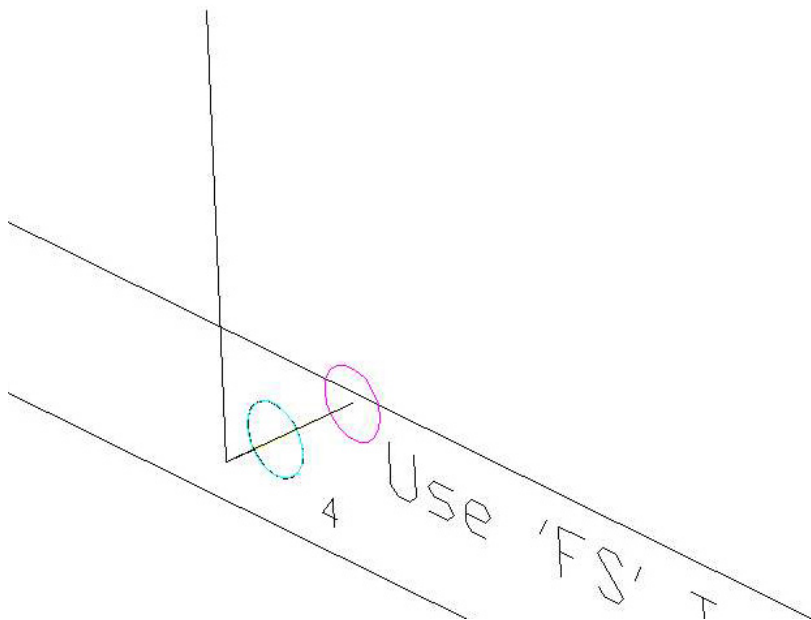
When using the Plane Detect option, Router-CIM will determine what plane the hole is being drilled on and will output a plane switch for the hole and offer another tool offset for the Horizontal Length. The tool path will look similar to FrontSide2, but the code will be very different.



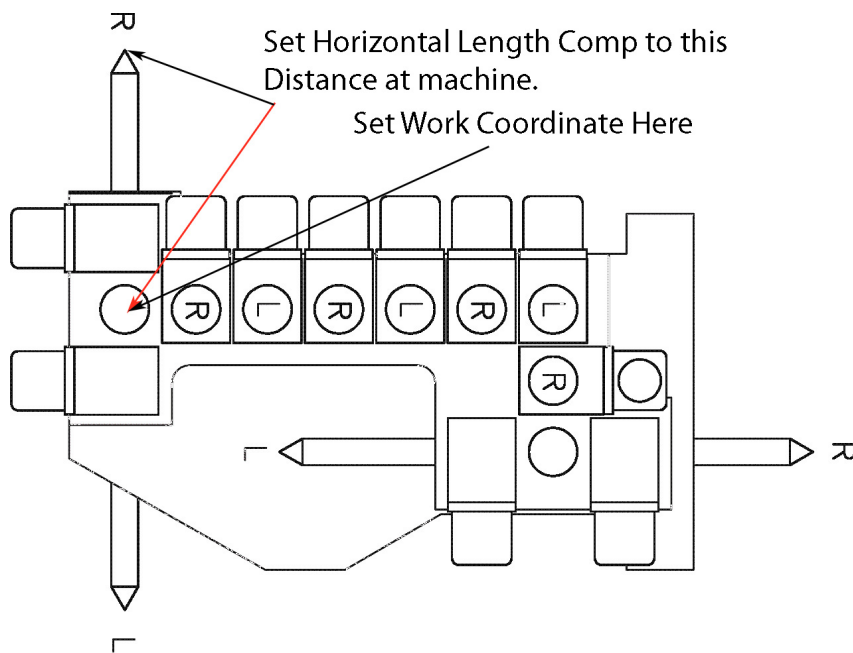
The setting of the Tool Length is now from the center of the spindle to the face of the collet that the drill fits into. The distance from the tip of the tool to the face of the collet can be put in at the machine into a regular offset. This way if the drill length changes, a new program does not have to be created, only the offset has to be updated on the machine and the code run again.

Tool Information DRILL-BIT_8MM Tools Tool Num / Comment "301" "DRILL 8MM - DIA." CRC Offset "01" Spindle Dir CW Tool Dia. 8.0 Tool Radius 4.0 Tool Length 150.0 4 Axis Safe 40.0 Type Category Vert. Offset Horiz. Offset Aggregate Offset Spindle Collet Cutter Compensation Yes No Both	Cycle Information Offset Dim 0 Cut Side inside Cut Direction CCW Round Corners n Lead In n Lead Out n Lead Size leadscl Lead Angle 0.0 Leadfeed Leadfeed Leadfeed Leadfeed	Status Information Safety Plane "6.35" Depth per Pass 25.4 Total Cut Depth -10.0 Feedrate/Spindle Speed Feedrate 2500.0 Spindle Speed 18000 Surface FPM NONE Units per Rev. NONE Calc Before Codes After Codes Oscillation Amt. 0.0000 Sort by Rank # DRILL MOTIONS 	Knowledge / Settings Knowledge Select Knowledge FRONTSIDE4 Doit Edit Edit Retrieve Save Import Export Tabbing No Yes Auto Tab @ Start Tab By Dist Tabbing Parameters Qty. NONE Length NONE Height NONE Dist. NONE MinRad. 0.0000 Acc-n-Dec Metric Plane Detect Inline Inverts Ramp Amt. NONE Overlap Amt. AUTO Doit File doitinfo.dat
---	---	--	--

Because the Aggregate Offset is set to Collet, the tool path looks normal:



The real trick here is to set the Horizontal offset correctly:



The code will look almost exactly like the previous sample, except that the X, Y values will be different because the tool path is in a different location.

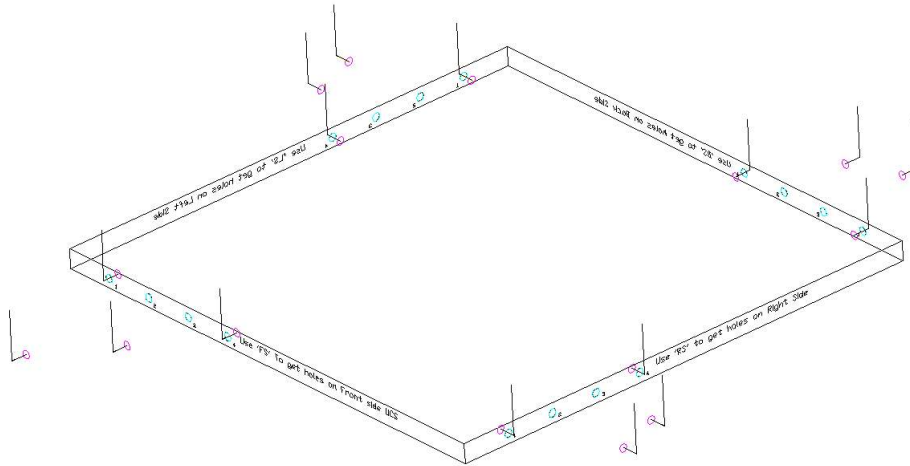
```
(DRILL 8MM - DIA.)
G28 G91 Z0 M05
G90 T301
G00 G17 G54P1 X200. Y-6.35 M03 S18000
G00 G43 H21 Z59.
G18 G44 H41
Z9.45
Y-2.54
G01 Y10. F2500.
G00 Y-6.35
G00 Z59.
H0
```

Using the Sample Drawing

There is a drawing name HorizontalBoringSample1.dwg which contains knowledge for each of the 4 cutting methods on each face. You can use Retrieve Knowledge to look at or export these sample knowledges into your own drawings.

You can download the drawing [here](#).

The sample has 4 holes drawn on each face, and one cut on each hole. Each shape is numbered to show which cut is which and there is text on each side to show which face you are working on.



The sample knowledges contained in the drawing are:

FrontSide1
FrontSide2
FrontSide3
FrontSide4
RightSide1
RightSide2
RightSide3
RightSide4
BackSide1
BackSide2
BackSide3
BackSide4
LeftSide1
LeftSide2
LeftSide3
LeftSide4

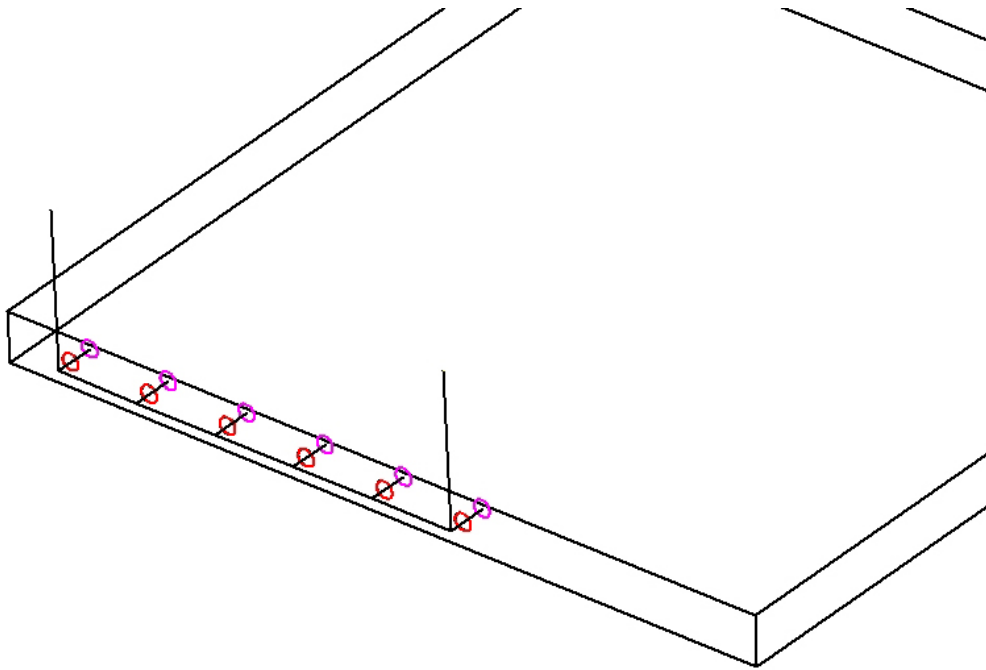
Avoiding Extra Safety Moves

Each time you make a cut on a horizontal plane, there is move from the safety plane, down to the depth of the cut in Z, then the cut is made in XY and the tool is moved back out of the hole and then back up to the safety plane in Z.

This means that if you were to drill several holes on the side of a shape, you would have the drill moving back up to the safety plane on each hole. This is mainly for safety as sometimes programs are stopped and restarted in locations that could cause a collision if the tool is left down at the hole depth.

If you want to avoid the extra moves in Z, you could simply group the holes on each side of the part after they are geoshaped and then drill them. This will create one Z move down and leave the head at that Z location and then move between each of the holes, returning to the Z height of the safety plane when the holes are done.

This is how the cycle would look when the holes are drilled:



This would work with either a horizontal drill in a drill block, or an aggregate tool drilling on the side of a part. If Plane Detect is turned on, only one horizontal comp move will be generated as well.

The code would look like this:

```
%
:1234(HBORING)
N1G00G17G20G28G40G80G91Z0M5
N2G90
N3G52X0Y0Z0
N4G08P1
N5M08
N6(DRILL .25 DIA.)
N7G53C90.
N8G28G91Z0M05
N9G90T2003M06
N10T102
N11M03S18000
N12G00G17G54P43X1.Y-.25C90.
N13G00G43H3Z2.
N14G18G44H83
N15Z-.375
```

```
N16G01Y.375F150.  
N17G00Y-.25  
N18X2.2598  
N19G01Y.375F150.  
N20G00Y-.25  
N21X3.5196  
N22G01Y.375F150.  
N23G00Y-.25  
N24X4.7794  
N25G01Y.375F150.  
N26G00Y-.25  
N27X6.0392  
N28G01Y.375F150.  
N29G00Y-.25  
N30X7.299  
N31G01Y.375F150.  
N32G00Y-.25  
N33G00Z2.  
N34H0  
N35G53C90.  
N36G28G91Z0M5  
N37G28G91X0M09  
N38G90  
N39G52X0Y0Z0  
N40G08P0  
N41M30  
%
```

Router-CIM Drawing Tools

Router-CIM Interactive has some built in AutoCAD drawing tools to assist in the creation of your parts. Select on each drawing option for more information:

- 1) [Creating a 'SLOT' in AutoCAD](#)
- 2) [Creating a 'Bubble Fillet' in AutoCAD](#)
- 3) [Using Router-CIM to change the UCS for Horizontal Drawing](#)

Creating a 'SLOT' in AutoCAD using Router-CIM

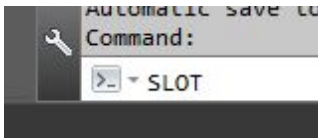
You can use Router-CIM within AutoCAD to draw the geometry of a SLOT in either the horizontal (aligned with the X axis in AutoCAD) or vertical (aligned with the Y axis in AutoCAD) direction. A Slot is defined as a rectangular piece of geometry with arcs on the end that match the width of the rectangle.

To use Router-CIM within AutoCAD to create what is defined as a SLOT, you will need to open AutoCAD and start Router-CIM within AutoCAD by selecting the Router-CIM icon under the toolbar or ribbon:



This will start Router-CIM within the AutoCAD drawing. If you are using Router-CIM Interactive, make sure to select the correct post processor from the Configuration Wizard. Select here for more information on the ['Configuration Wizard'](#).

Once Router-CIM has been successfully started within the AutoCAD drawing, you can then type in the command line SLOT.



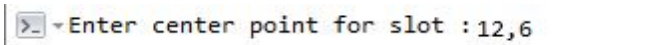
This will start the SLOT command.

NOTE: If you type in SLOT and it comes up with Unknown Command, it means that Router-CIM was not started in that AutoCAD drawing.

The following information was based on a slot that was drawn centered on a 24 x 12 rectangle that was drawn from 0,0 in AutoCAD in Quadrant 1.

When the command is active, you will be prompted to follow a series of questions.

The first question is asking for the center point of the slot:

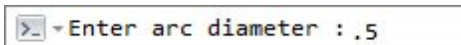


The value that should be entered is either based on a button click or defining the X and Y coordinate.

The next question will ask for the distance between the two centers of the arcs. You can also think of this as the length of the rectangular feature:



The next question will ask for the DIAMETER of the arcs that are at the ends of the rectangle. You can think of this as the width of the rectangle.



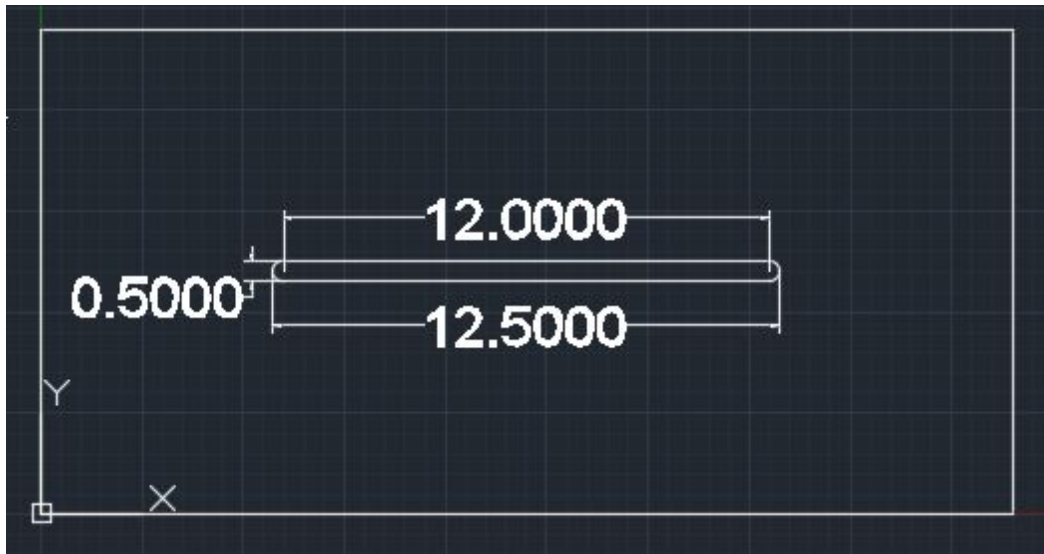
The final question will be whether to draw the SLOT aligned with the X axis in AutoCAD, Horizontal (H) or should it be aligned with the Y axis, Vertical (V).



Once this final question is completed, Router-CIM will draw a SLOT according to the information given.

NOTE: A SLOT will be drawn as 2 lines and 2 arcs that are NOT joined.

Here is an example of a SLOT drawn according to the information defined above:



Creating a 'BUBBLE FILLET' in AutoCAD using Router-CIM

You can use Router-CIM within AutoCAD to draw the geometry of a BUBBLE FILLET. A BUBBLE FILLET is defined as a rectangular piece of geometry with arcs on the end that match the width of the rectangle.

To use Router-CIM within AutoCAD to create what is defined as a BUBBLE FILLET, you will need to open AutoCAD and start Router-CIM within AutoCAD by selecting the Router-CIM icon under the toolbar or ribbon:



This will start Router-CIM within the AutoCAD drawing. If you are using Router-CIM Interactive, make sure to select the correct post processor from the Configuration Wizard. Select here for more information on the ['Configuration Wizard'](#).

Once Router-CIM has been successfully started within the AutoCAD drawing, you can then type in the command line BFLT or select the icon from the toolbar or ribbon.



This will start the BUBBLE FILLET command.

NOTE: If you type in or select the icon for BUBBLE FILLET and it comes up with Unknown Command, it means that Router-CIM was not started in that AutoCAD drawing.

The following information was based using the Router-CIM Bubble Fillet command on an inside rectangular feature.

When the command is active, you will be prompted to follow a series of questions.

The first question will ask for the fillet radius that you want:

A screenshot of the AutoCAD command line showing the prompt '> Fillet radius: <0.5000000>'.

The value that should at least the radius of the tool that you will be using to accomplish this operation. The default that will come in is 0.5.

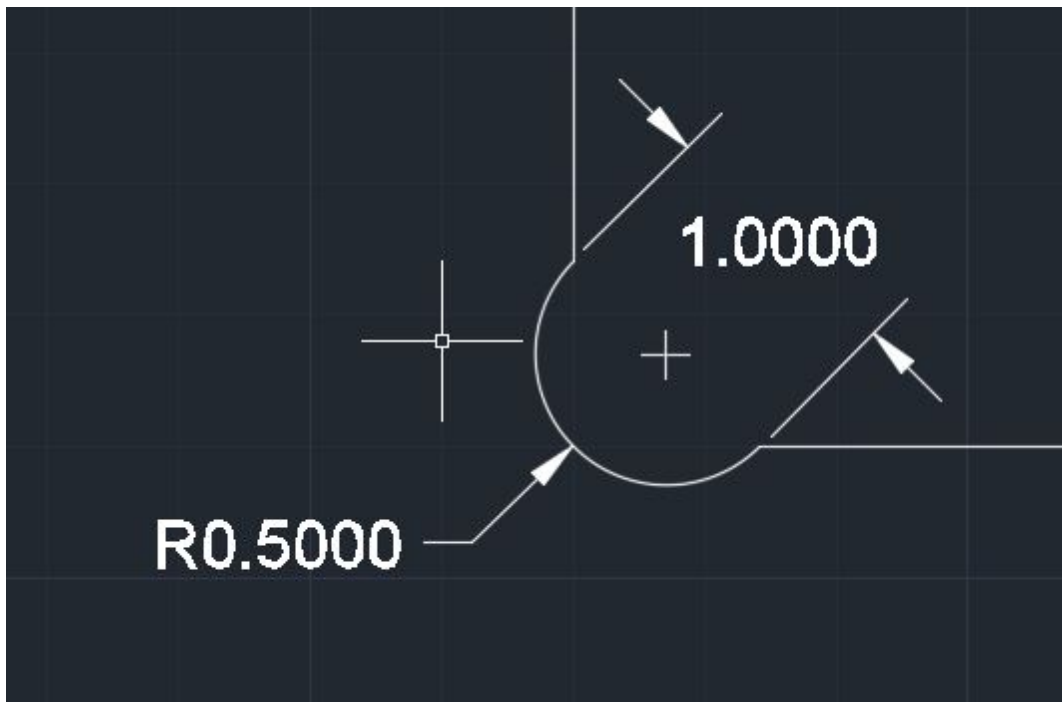
The next question will ask for the first object. This will be the first piece of geometry of an adjacent feature:

A screenshot of the AutoCAD command line showing the prompt '> Select first object:'.

The next question will ask for the second object. This will be the second piece of geometry that is adjacent to your first object.

A screenshot of the AutoCAD command line showing the prompt '> Select second object:'.

Based on this information, a BUBBLE FILLET will be drawn where the first object and second object intersect as shown below:



Converting a 2D to 3D Polyline

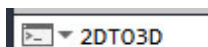
When you are drawing on a UCS that is outside the World Coordinate System (WCS) that you would like to cut with a 2D Router-CIM cutting cycle. The 2D geometry will need to be converted to a 3D polyline. Router-CIM includes a command called 2Dto3D to assist in converting the 2D geometry.

To use Router-CIM within AutoCAD to convert a 2D piece of geometry to a 3D polyline, you will need to open AutoCAD and start Router-CIM within AutoCAD by selecting the Router-CIM icon under the toolbar or ribbon:



This will start Router-CIM within the AutoCAD drawing. If you are using Router-CIM Interactive, make sure to select the correct post processor from the Configuration Wizard. Select [here](#) for more information on the '[Configuration Wizard](#)'.

Once Router-CIM has been successfully started within the AutoCAD drawing, you can then type in the command line 2Dto3D.



This will start the 2Dto3D command.

NOTE: If you type in 2Dto3D and it comes up with Unknown Command, it means that Router-CIM was not started in that AutoCAD drawing.

With the command active, simply select on the 2D geometry and Router-CIM will convert it to a 3D polyline.

Router-CIM Automation Suite

The main purpose of Router-CIM Automation Suite is to provide a complete CAD/CAM solution for both single part machining and nested based manufacturing. Router-CIM Automation Suite allows you to automatically produce NC Code for parts that are produced or imported into the AutoCAD environment, as well as any other single part that can be parametrically defined, or produced as a drawing or DXF file.

For new users of Router-CIM Automation Suite, there are a few concepts which need to be understood in order to use the product efficiently. These are:

- A basic understanding of the Microsoft Windows environment, and file handling with programs such as My Computer, or Windows Explorer.
- A working knowledge of the AutoCAD environment. How to create and name layers, and assign geometry to those layers.
- Some knowledge of the types of tools, cutting conditions, and materials you are likely to use on your parts.
- Knowing the difference between the file types Router-CIM Automation Suite uses such as DWG, DXF, and SCN.
- Understanding the Layer to Knowledge association feature in Router-CIM Automation Suite.

If you will be primarily using macros for your part definitions, then a complete understanding of Router-CIM's Parametric Macro Builder and how to create a macro is truly necessary. This takes practice and is best done on paper first! Also you will need to understand the different variable types: Global, Dynamic, Tagged, and Local, and how to use them.

The Router-CIM Automation Suite is a comprehensive suite of tools that allows you to process any number of parts without user intervention. The capabilities include (but are not limited to):

- Cutting any number of parts with a maximum of 1999 unique parts
- Nesting
- Sorting
- Automated NC Code Creation
- Common Line Cutting (Advanced Nesting Module)
- Tool Stay Down Cutting (Advanced Nesting Module)
- Labeling
- Reporting

The basic premise for the entire Router-CIM Automation Suite is Layer to Knowledge processing using the DOIT cycle. This provides for tool paths to be created in a standardized, regulated, and sortable manner for any type of part.

The following sections are broken up into the different areas of the Router-CIM Automation Suite program and are designed to give you as much information about the tools available as possible.

This section is meant to be a reference to the Router-CIM Automation Suite, and not a tutorial. Tutorials will be provided in other areas as they are created.

Starting Router-CIM Automation Suite

To start Router-CIM Automation Suite, simply double click on the icon that should be located on your desktop.

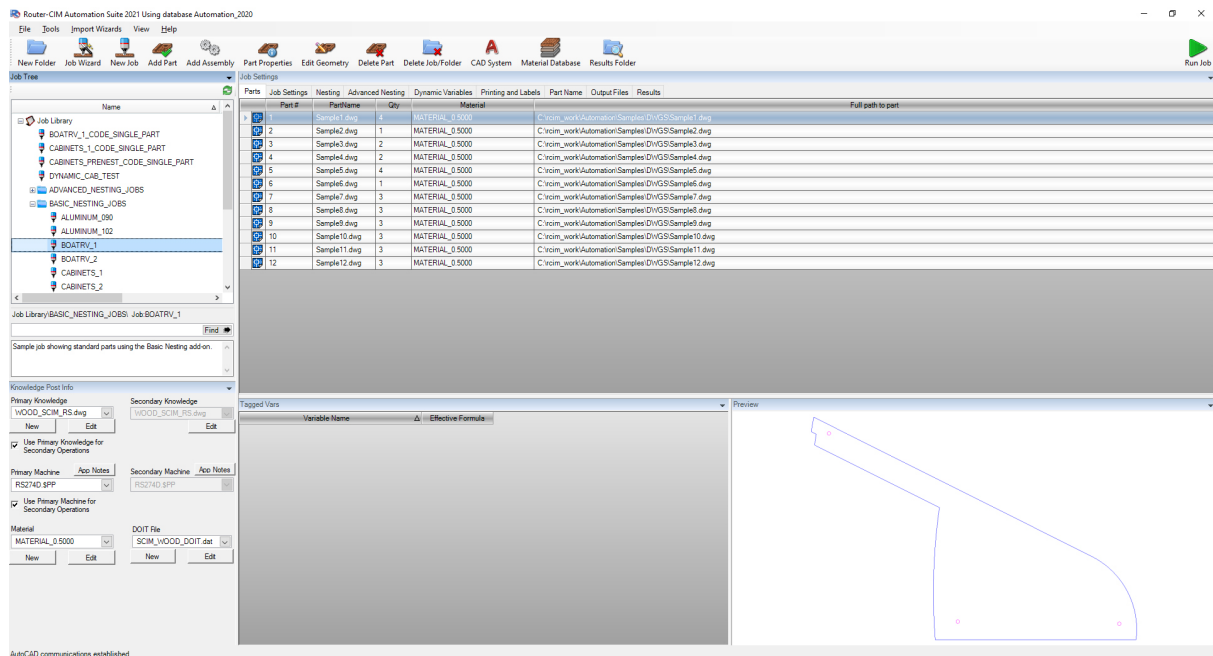


You can also start Router-CIM Automation Suite through your Apps in Windows 10 or through your programs in Windows 7.

The Automation Interface

The Automation Interface is the link between you and Router-CIM Automation Suite. All the settings within Router-CIM Automation Suite are presented so that jobs can be made in either a completely repetitive manner, or a completely custom manner from job to job.

The interface is broken down into several parts. Each of these parts has a chapter describing its functionality. You can select on the items in the picture below for a link to each chapter.



- 1 - [Menu Area](#)
- 2 - [Toolbar](#)
- 3 - [Folder Tree](#)
- 4 - [Part Window](#)
- 5 - [Variables Window](#)
- 6 - [Knowledge Settings](#)

Menu Area

The Menu Area of Router-CIM Automation Suite contains the following sections:

File

- [Database Maintenance](#)
- [Settings](#)
- [Print Options](#)
- [Exit](#)

Tools

- [Create New Knowledge Drawing](#)
- [Create Pack and Go](#)
- [Clear AutoNest Task and Part Folders](#)
- [Clear ShapeDone Folder](#)
- [Restart AutoCAD UI Engine](#)
- [Restore Startup Drawing](#)
- [Save Current Settings as Job Defaults](#)
- [Automatic G-Code Scan](#)
- [Migration Wizard](#)
- [Multi Post Defaults](#)
- [NCVar Editor](#)
- [RCIM Labeling](#)
- [Single Part Report Configuration](#)

Import Wizards

- [Excel and Comma Delimited \(CSV\) Import Wizard](#)
- [Solid-CIM CSV Import](#)
- [Pack and Go Import](#)
- [Select and Go Import](#)
- [Additional Imports](#)

View

- [Show Job Tree](#)
- [Show Job Settings](#)
- [Show Knowledge Post Info](#)
- [Show Tagged Vars](#)
- [Show Part Preview](#)
- [Save Current View](#)
- [Reset to Saved View](#)
- [Reset to Default Automation View](#)

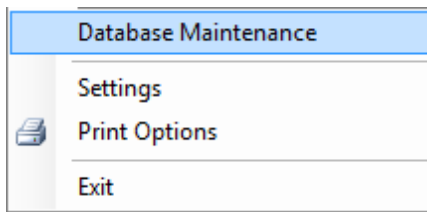
Help

- [Help](#)
- [About](#)

File Menu

There are several options available in the File menu. These options include database management, Automation Suite settings, and options for selecting your default printer for Automation.

Each of the items will be explained in detail in the following sections.



Database Maintenance

Database Maintenance is a set of utilities for working with the SQL databases in Router-CIM Automation Suite. From these menu choices you can back up, restore or even convert an old database to the new format.

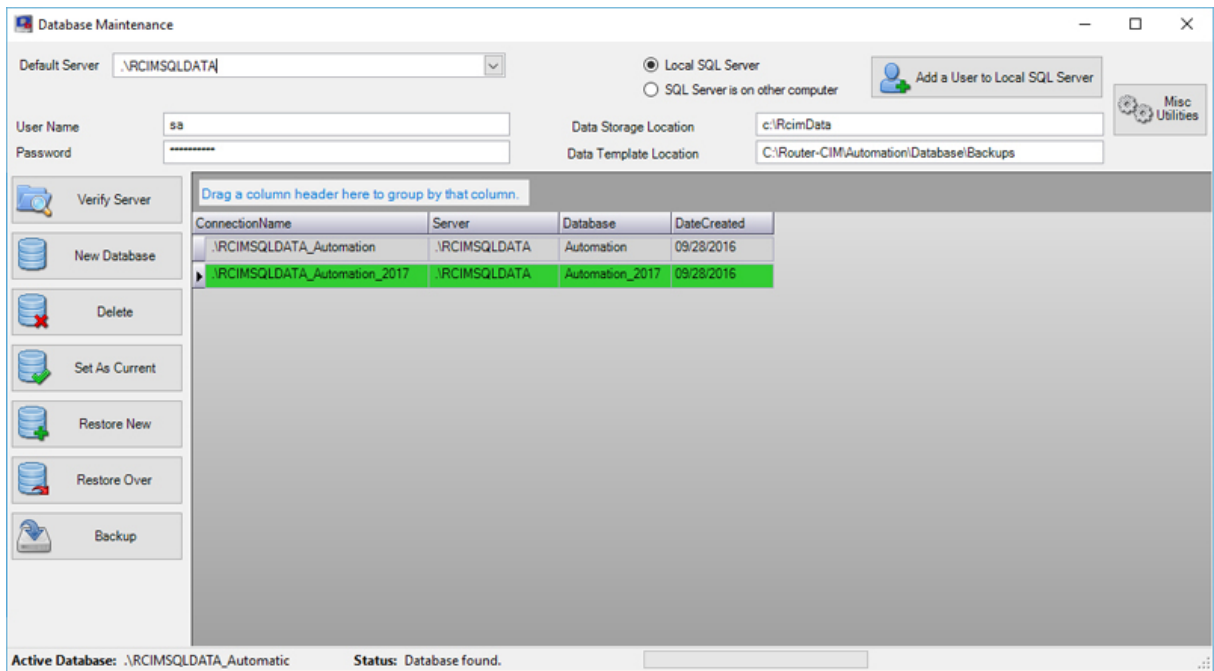
The Database Manager can also be accessed from the desktop by double clicking on this icon:



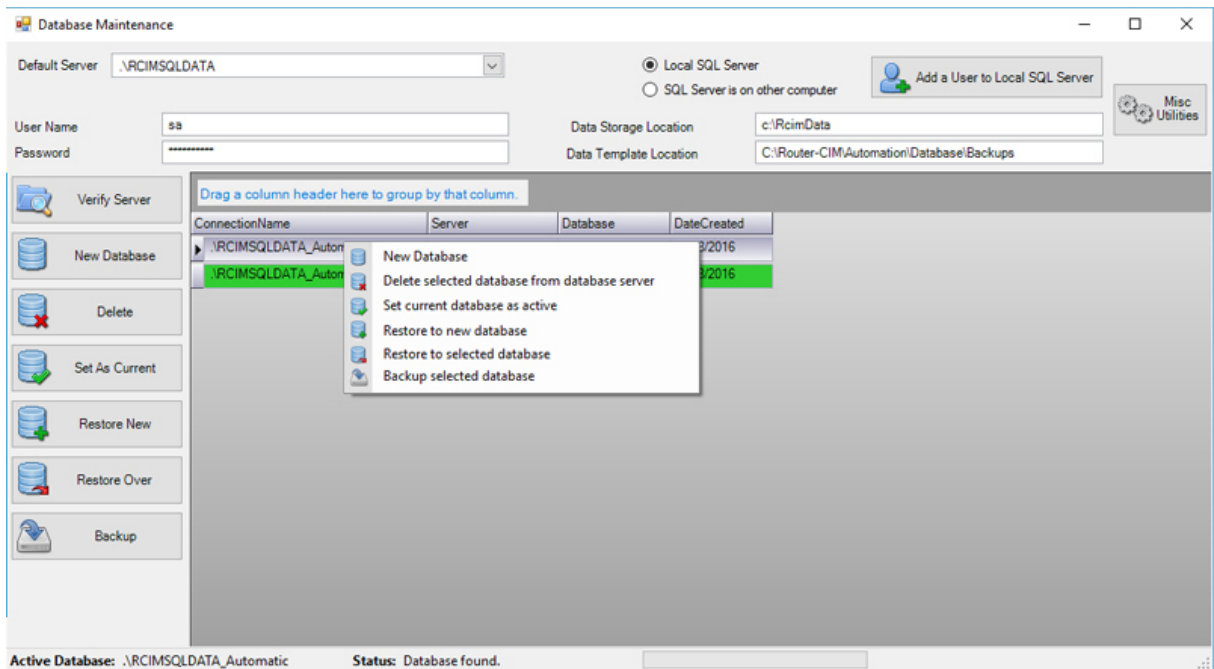
The choices available are

- [Verify Server](#)
- [New Database](#)
- [Delete](#)
- [Set as Current](#)
- [Restore New](#)
- [Restore Over](#)
- [Backup](#)
- [Misc. Utilities](#)
- [Shared Databases](#)

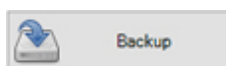
Selecting the Database Maintenance will show the Database Utilities window where all the options for modifying and creating your databases are kept.



Right-clicking on the databases in the list window will show choices available.

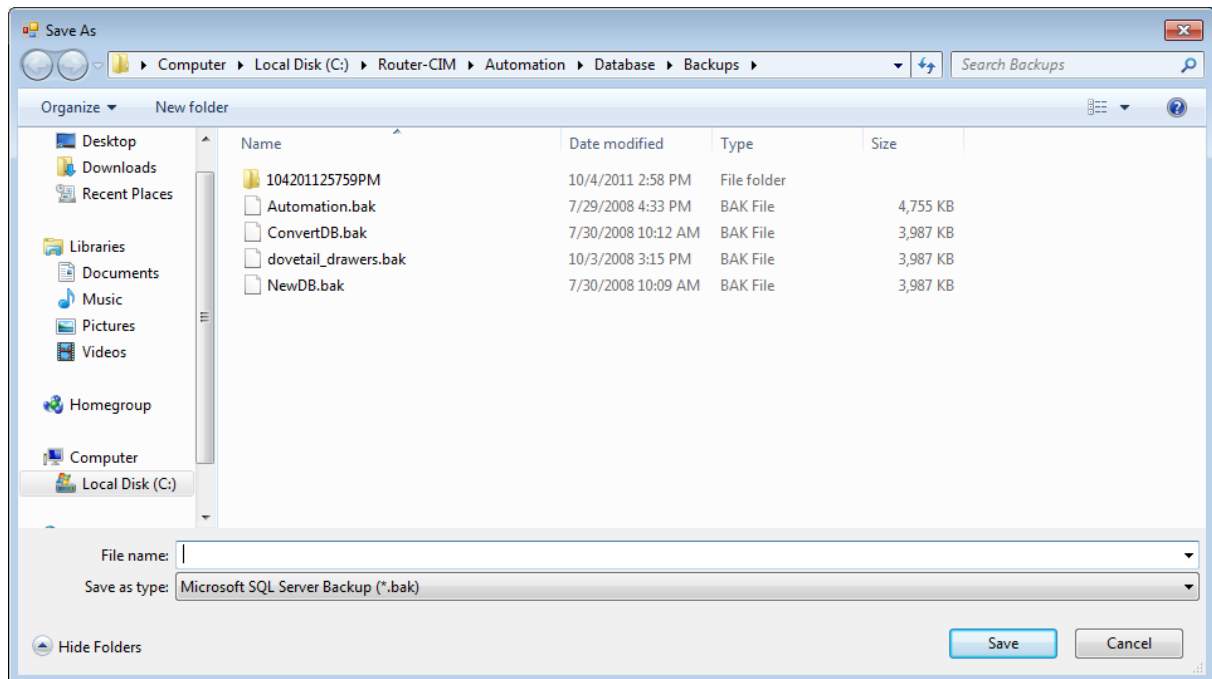


Backup Database

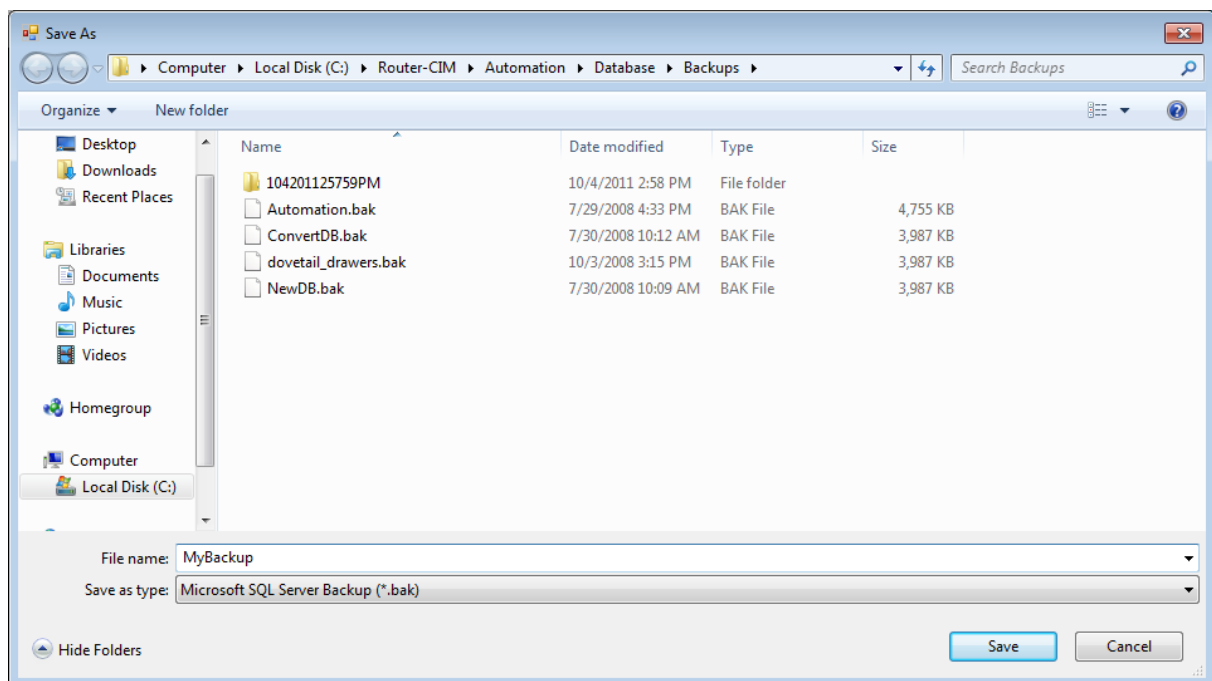


The Backup function will back up the selected database to a backup file format for later re-use with the Restore command.

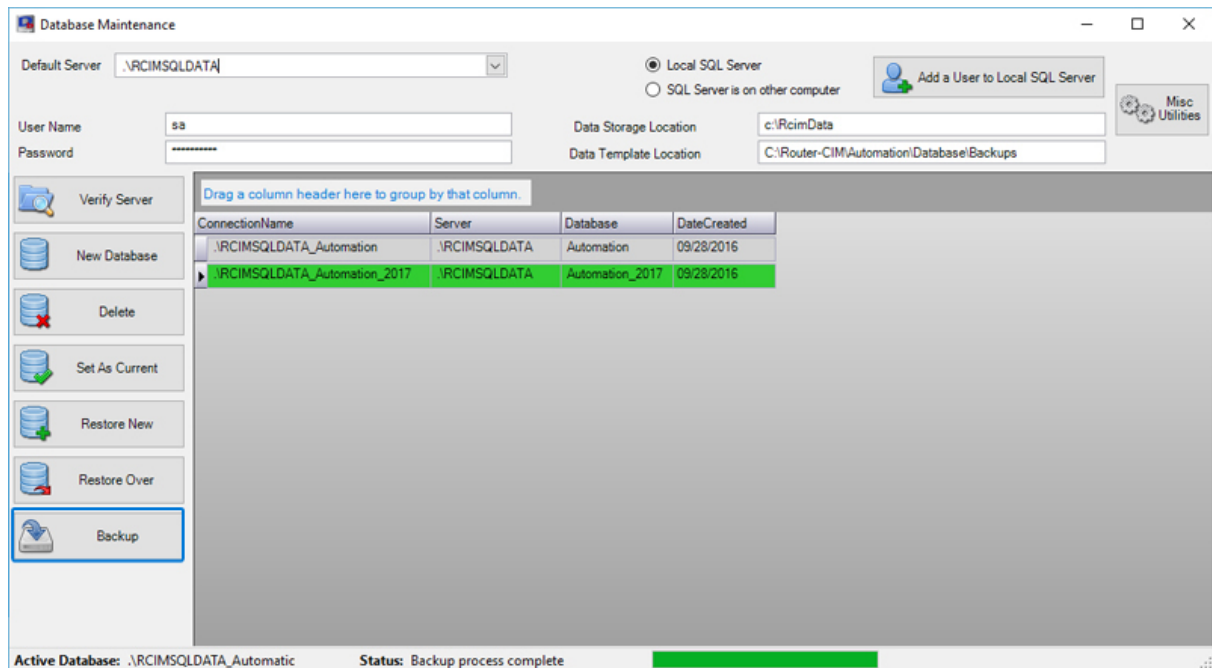
Once you select a database, and select the **'Backup'** button (or Right Click on a database and select Backup Selected Database) a window will open looking for a name to back up the database to.



You have to enter a name for the backup file for the database.



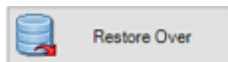
Type a name into the File Name field and then select **'Save'**.



Once you select Save, a backup of the database will occur, and a file with the name of the file and a .BAK extension will be created.

The Default Folder for a database backup is C:\Router-CIM\Automation\Database\Backups.

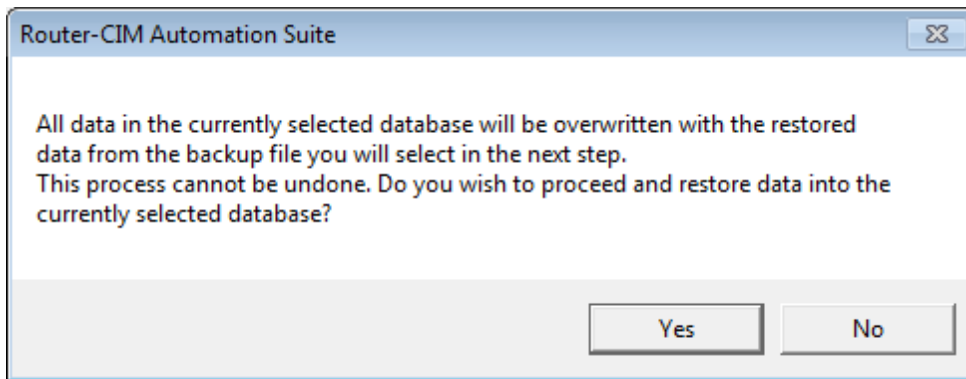
Restore Over Database



Using Restore Over will reconstruct a database that you had used previously and that was stored as a backup in the Router-CIM Automation Suite system, over the top of the currently selected database. This option completely overwrites the current database with the data from the backup file you select.

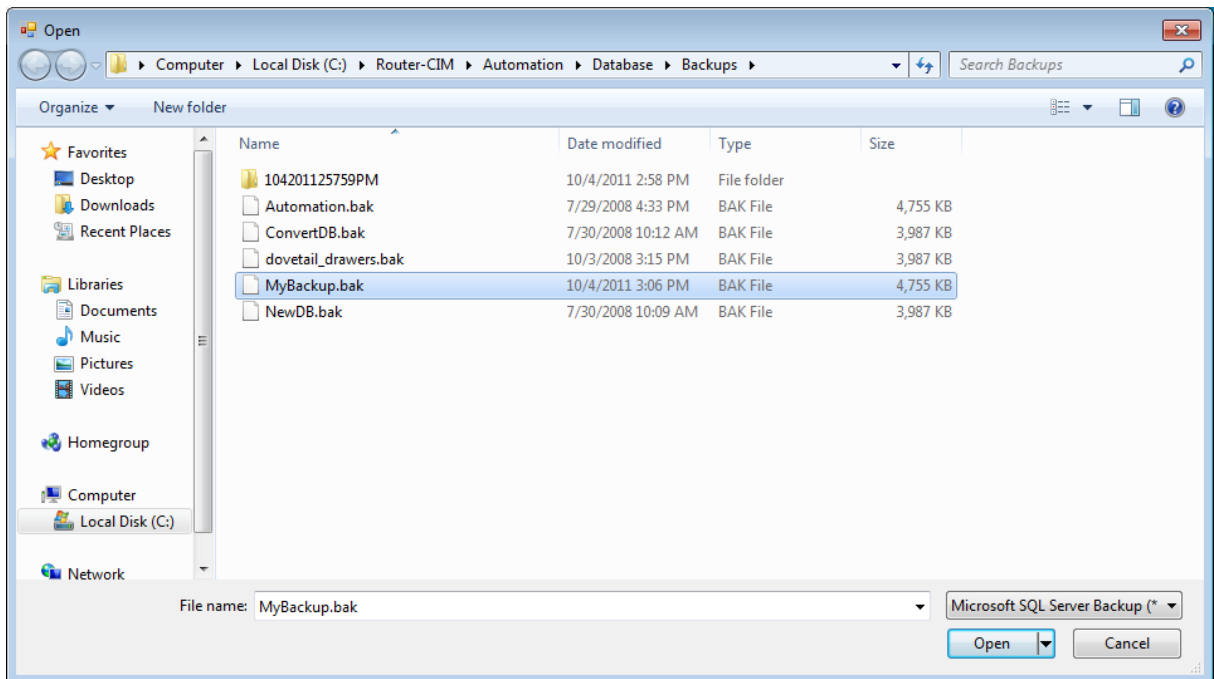
Note: In order to restore a backup database, the .BAK file must be located in the C:\Router-CIM\Automation\Database\Backup folder.

Once you select '**Restore Over**', you will have to confirm that you want to overwrite the currently selected database with the backup version you select in the next step.

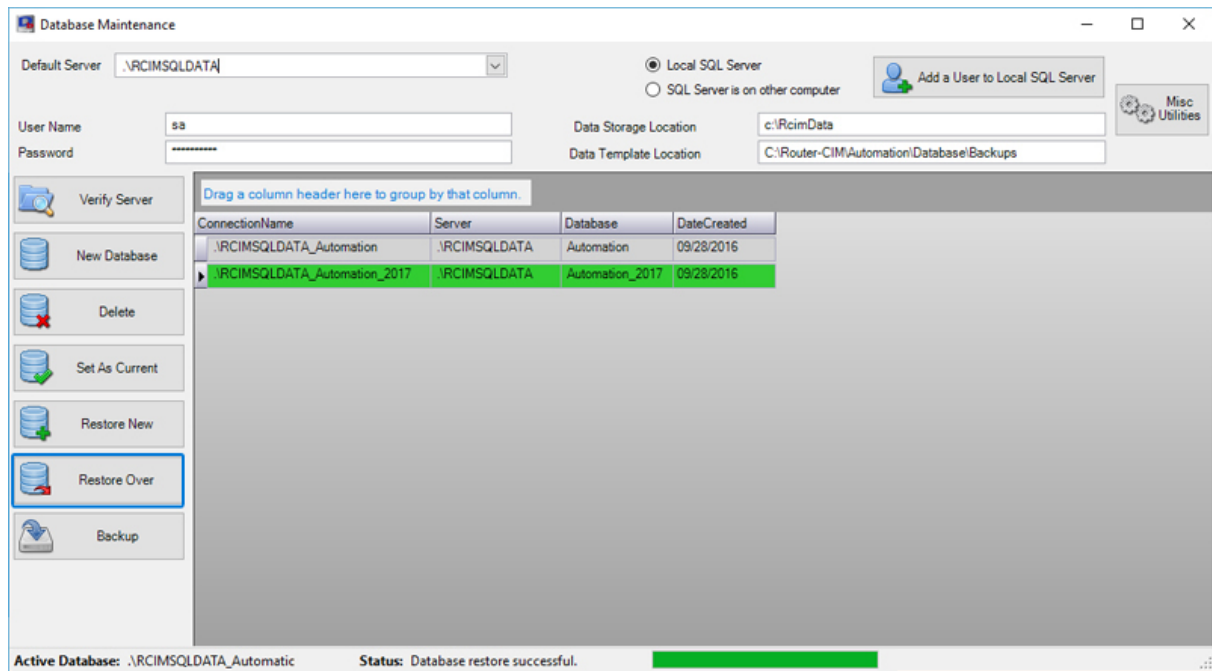


Select '**Yes**' to restore over the currently selected database.

A new dialog box will open for you to select the database you want to restore into the currently selected one.

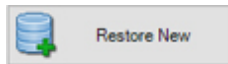


Select the database you want to restore and then select '**Open**'.



After selecting 'OK', you will see the database window, while the restore is in process.

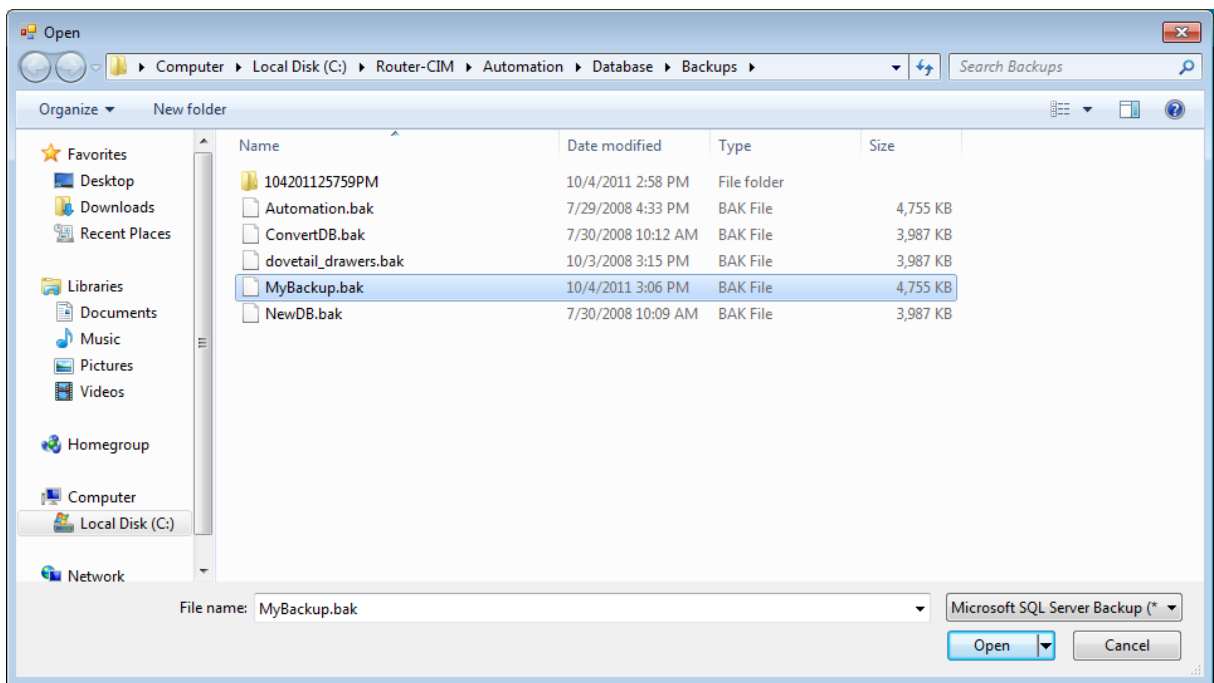
Restore New Database



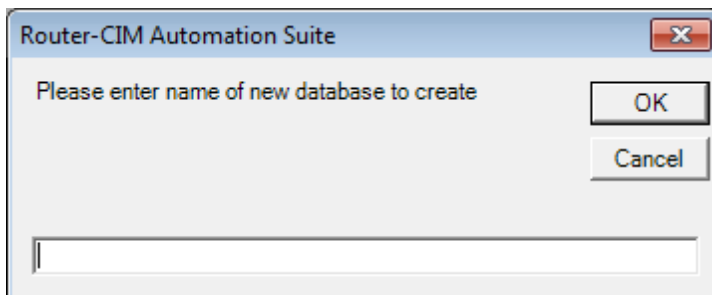
Using Restore New will reconstruct a database that you had used previously and that was stored as a backup in the Router-CIM Automation Suite system, in a new database. This creates a new database with the data from the backup file you select.

Note: In order to restore a backup database, the .BAK file must be located in the C:/Router-CIM/Automation/Database/Backup folder.

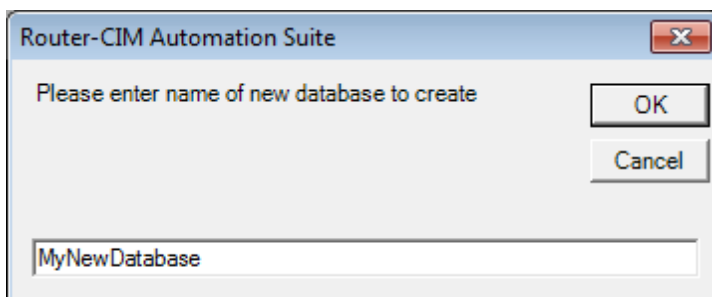
Once you select Restore New, you will have to select the database you want to restore.



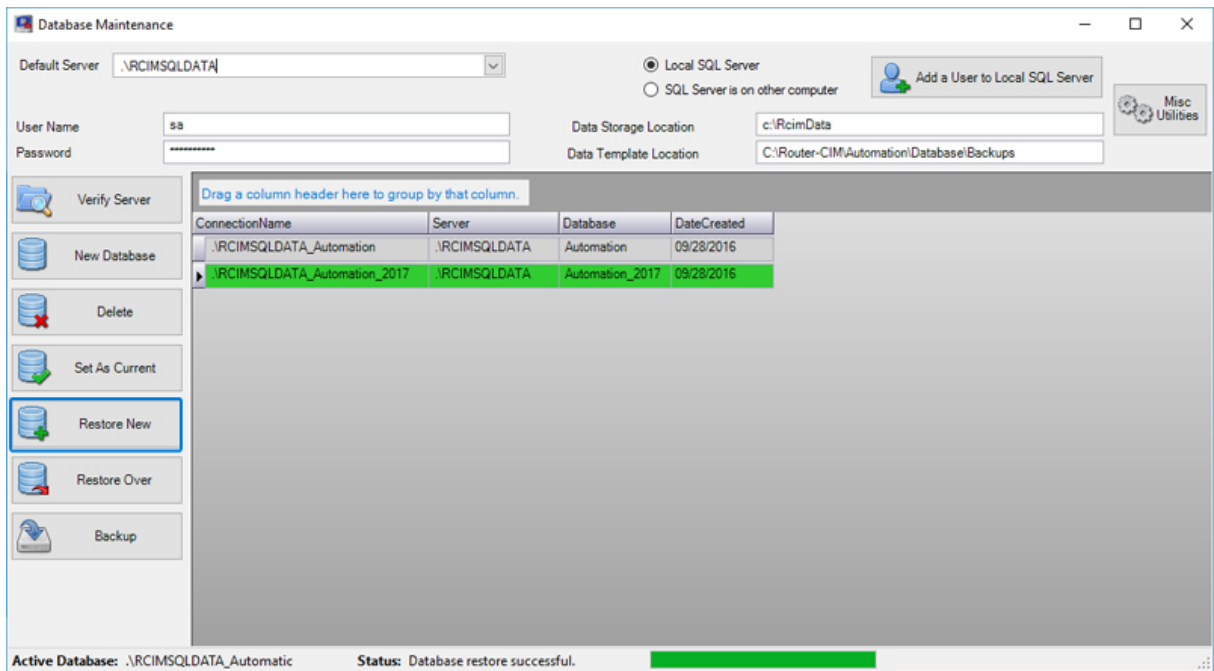
Then you will enter a name for the database you want created from the new process.



Type in a name for the new database.

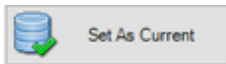


Then select 'OK'.



After selecting 'OK', you will see the database window, while the restore is in process.

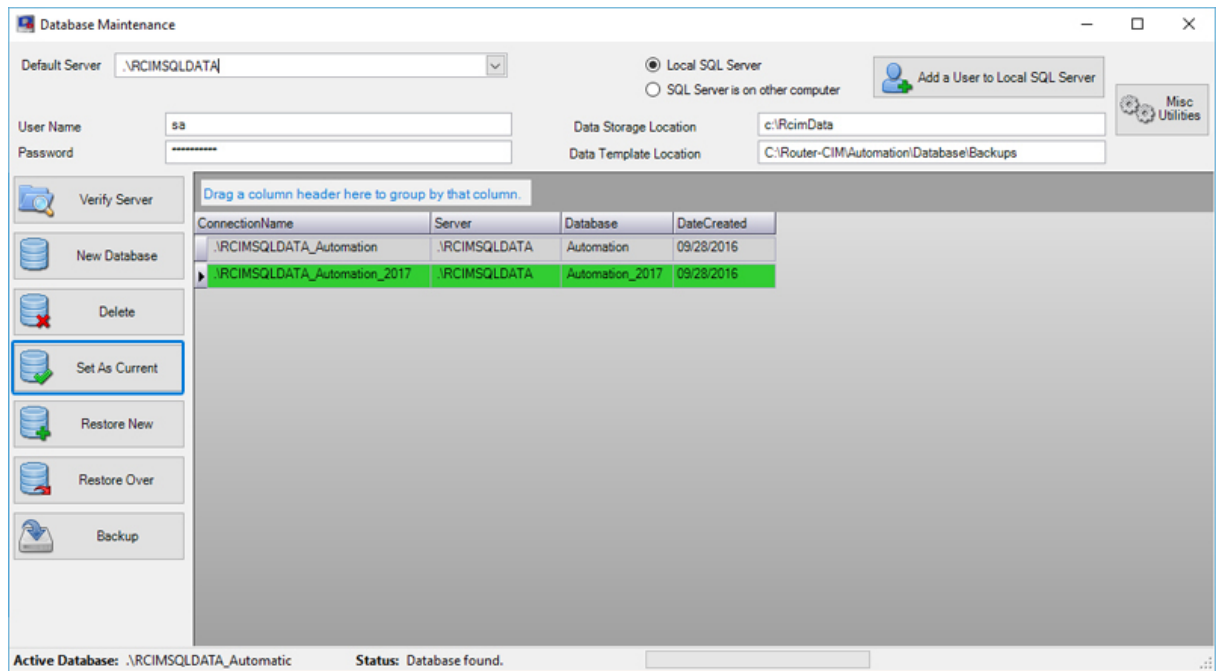
Set as Current



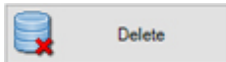
Using 'Set as Current' will change the database Router-CIM Automation Suite is currently using to the database selected and make that database current.

The currently selected database will be highlighted in green.

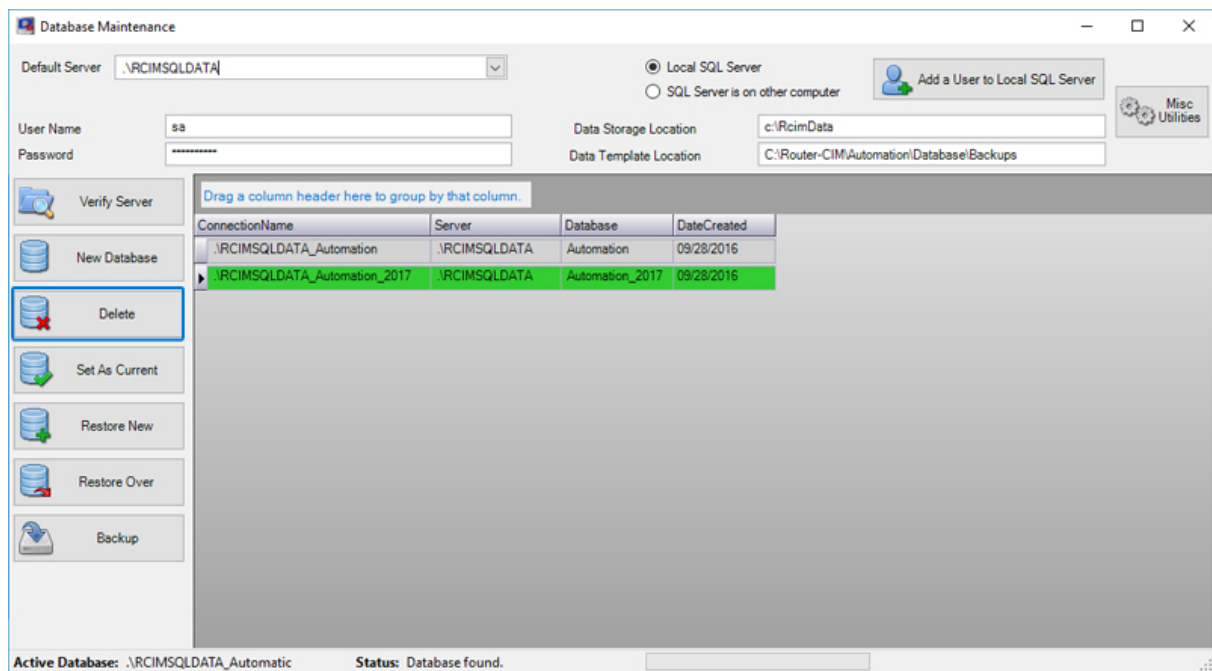
To change the current database that Router-CIM Automation Suite uses, select the correct database and then select the 'Set As Current' button.



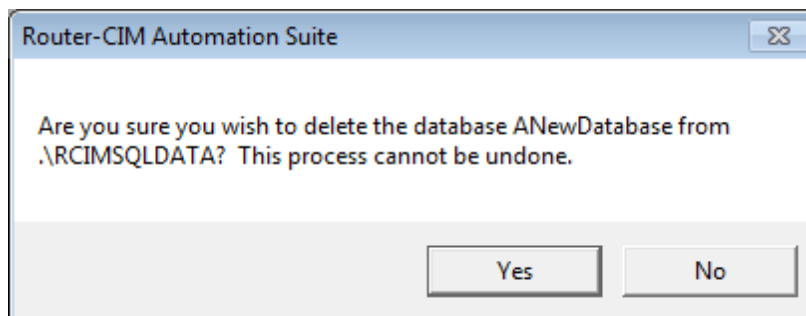
Delete Database



Delete a database from the current list of available Databases.

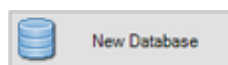


You will be prompted to be sure you want the database deleted.



Once deleted, the database cannot be restored (except from a backup file if it exists).

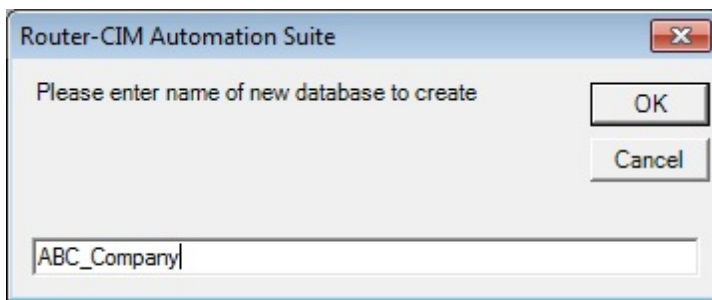
New (Database)



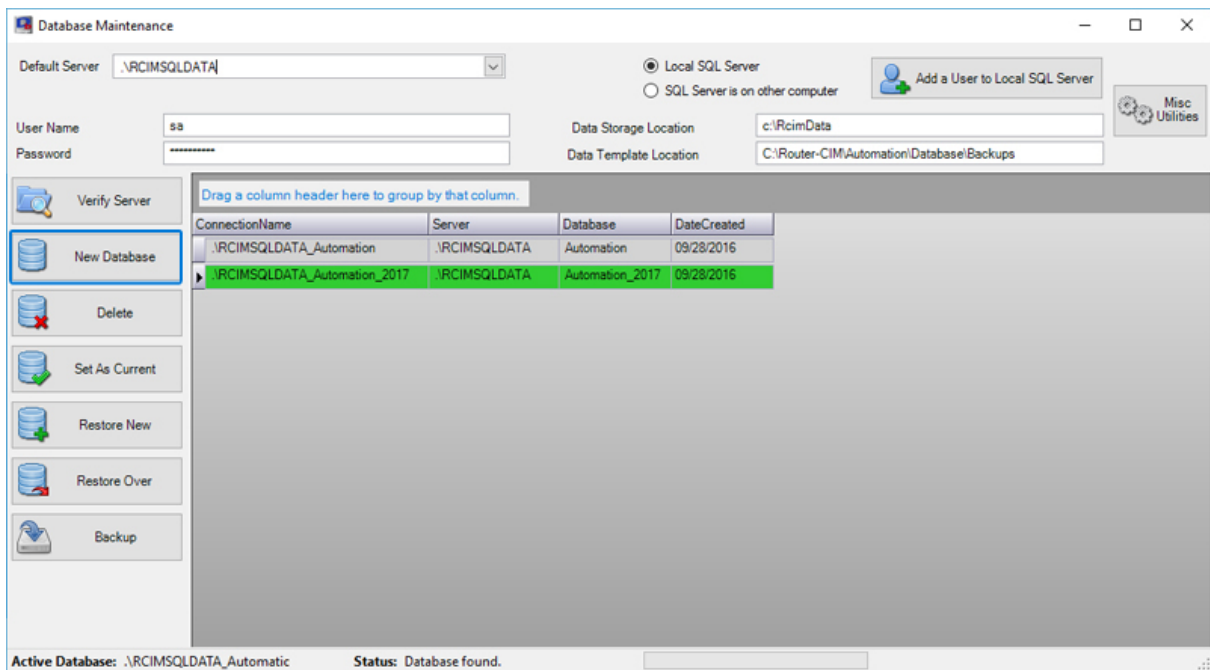
Selecting **New Database** from Database Maintenance will allow you to create a new database entry in the database list and will allow you to make that database current.

This selection is used when you wish to create a new, blank database to work from.

The first window will prompt you for the name of your new database.



Type in a name and select '**OK**'.



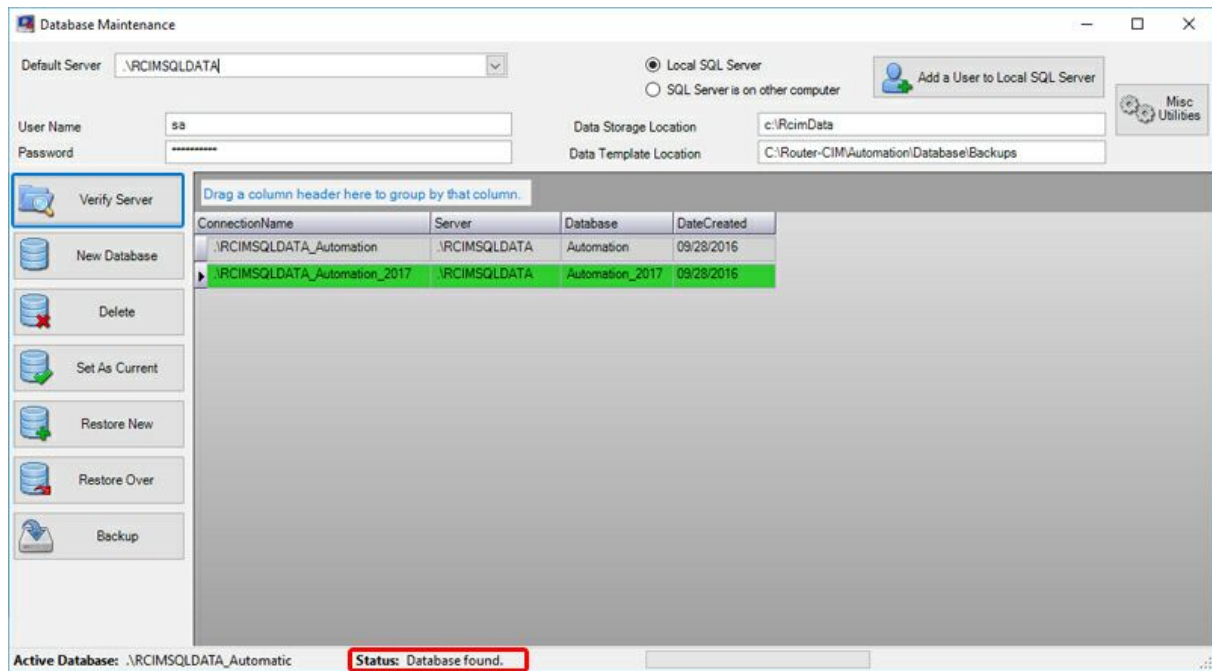
Make sure to select the '[Select As Current](#)' button if you wish to make this the active database that is used by Router-CIM Automation Suite.

Verify Server

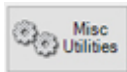


Selecting **Verify Server** will allow Router-CIM Automation Suite to check and see whether or not the database server (service) is running on your computer.

A notice at the bottom of the Database Maintenance screen should show **Status: Database found**.



Misc. Utilities



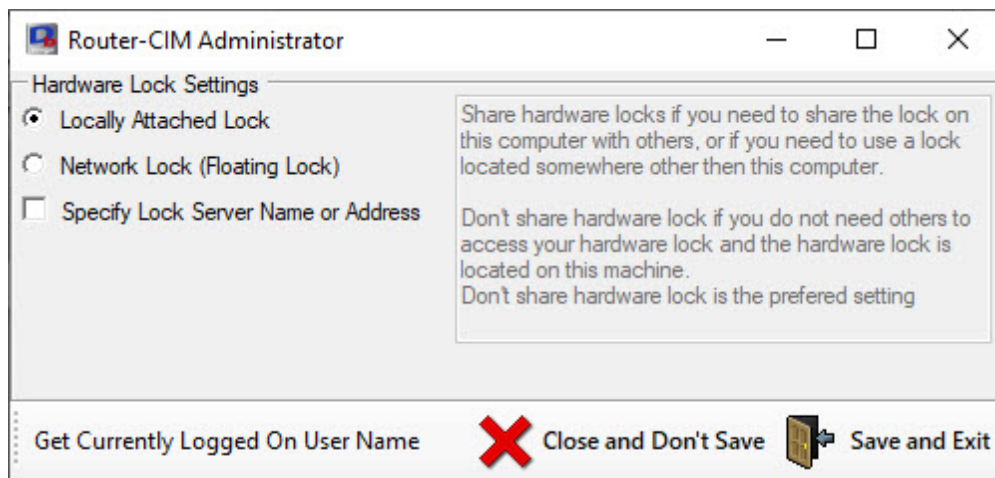
The Misc. Utilities Function will allow you to set the following item:

- Locally Attached Lock - The Router-CIM hardware lock needs to be inserted into the current users computer directly
- Network Lock (Floating Lock) - Allow you to store the Router-CIM hardware lock on a server or a different computer.

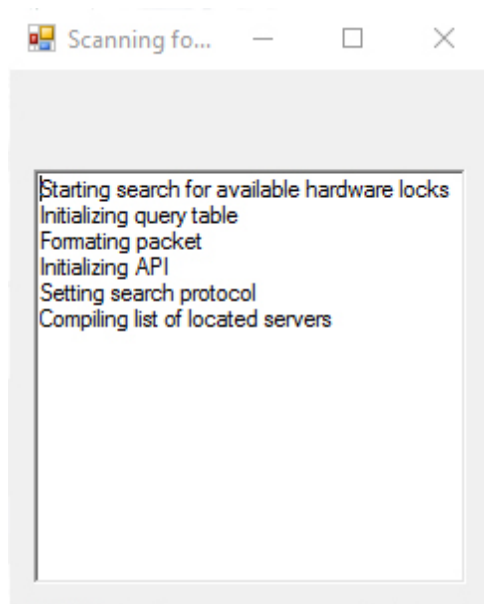
If Network Lock (Floating Lock) is selected, you can select the 'Specify Lock Server Name or Address' option and identify the server name or IP address in order to allow the user to quickly locate the Router-CIM hardware lock.

Note: This option applies to the Router-CIM hardware lock and does not apply to the Nesting lock if the option was purchased.

Note: Type 1 Hypervisor Appliance Compatibility Warning: Please be aware that neither Router-CIM, Solid-CIM or Radan's Autonest USB hardware locks are compatible with any Type 1 Hypervisor appliance, including, but not limited to, Microsoft Hyper-V, Oracle VM, KVM and Citrix Hypervisor. These products ARE compatible with Type 2 Hypervisor appliances, such as VMWare's VM Workstation.



If the 'Network Lock (Floating Lock)' is selected, when you start Router-CIM or a component that requires the Router-CIM hardware lock, you will see this dialog box letting you know that it is searching for the lock:



If it does not find the Router-CIM hardware lock or the lock is currently in use, it will notify you.

Setting up a Shared Database

Setting up a Shared Database allows you to have multiple users working from the same jobs, settings and materials in Router-CIM Automation Suite. This will assist in keeping multiple users producing the same results through Router-CIM Automation Suite.

There are two methods for setting up Router-CIM Automation Suite with a shared database:

- 1) Selecting a Master Computer (complete installation of AutoCAD and Router-CIM required) that all computers have access to.

2) Setting up the SQL Server on a stand-alone server (AutoCAD and Router-CIM installation on the server is not required). Requires additional support files to be downloaded. **Please proceed to the ['Installing on a Stand-Alone Server'](#) section before setting up the shared database.**

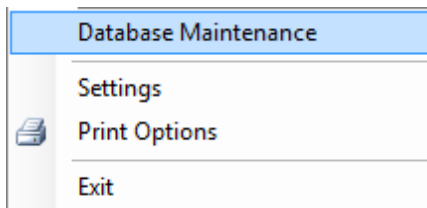
Either process will be setup the same way following these instructions:

Note: Setting up a shared database using a master computer assumes that all computers being setup with have access to the master computer at all times when running Router-CIM Automation Suite.

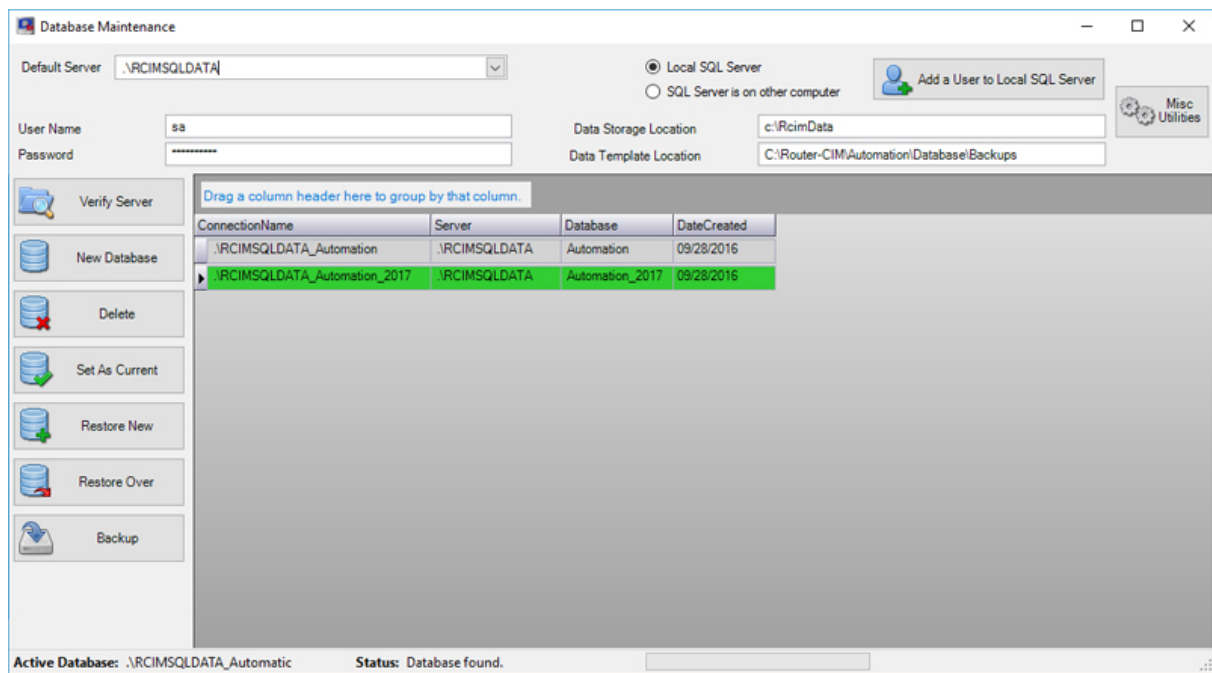
Note: Setting up a shared database using a stand-alone server assumes that all computers being setup with have access to the stand-alone server at all times when running Router-CIM Automation Suite.

Note: These procedures ONLY need to be done on the client computers, when setting up Shared Databases using a Master Computer. The Master Computer will need no changes.

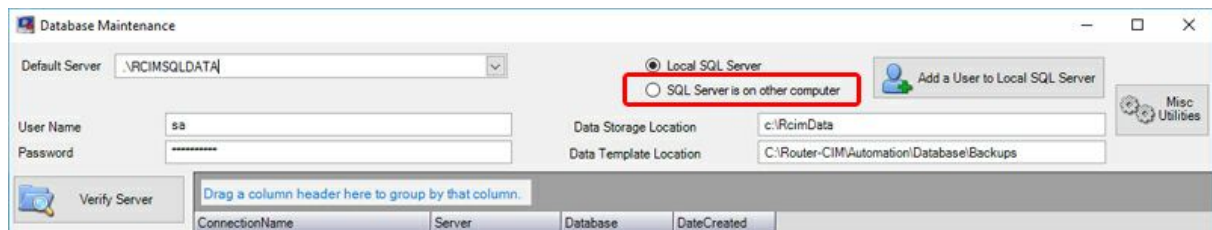
On a client computer, select '**Database Maintenance**' under the '**File**' menu in Router-CIM Automation Suite:



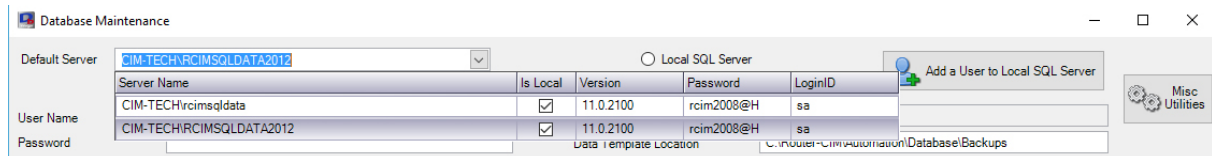
Once selected, the Database Manager will open:



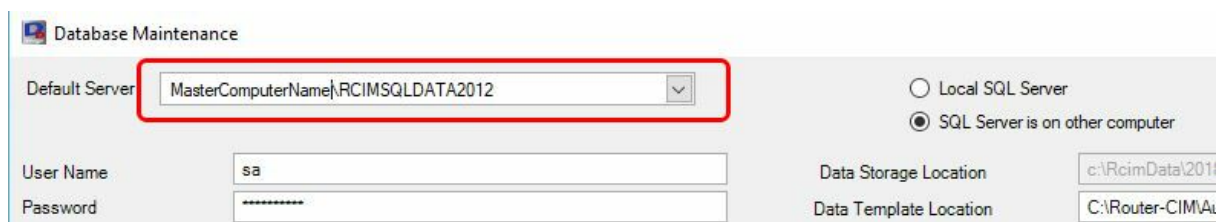
With the Database Manager open, select the option on the top of the window for 'SQL Server is on other computer':



You can then proceed to the drop-down option under 'Default Server'. This will show you the SQL Servers that your computer has access to:



If you do not see the SQL Server that you are trying to access on the Master Computer, you can manually type in the name of the Master Computer followed by the correct database name:



Database Maintenance

Default Server: MasterComputerName\RCIMSQLDATA2012

User Name: sa

Password: *****

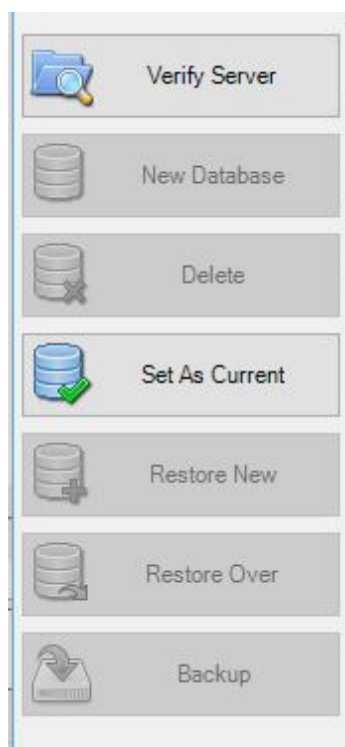
Local SQL Server: ☐

SQL Server is on other computer: ☒

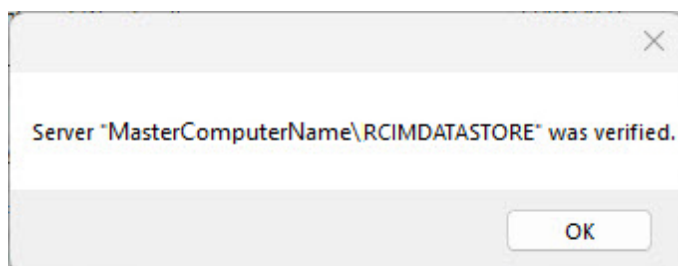
Data Storage Location: c:\RcimData\2012

Data Template Location: C:\Router-CIM\Au

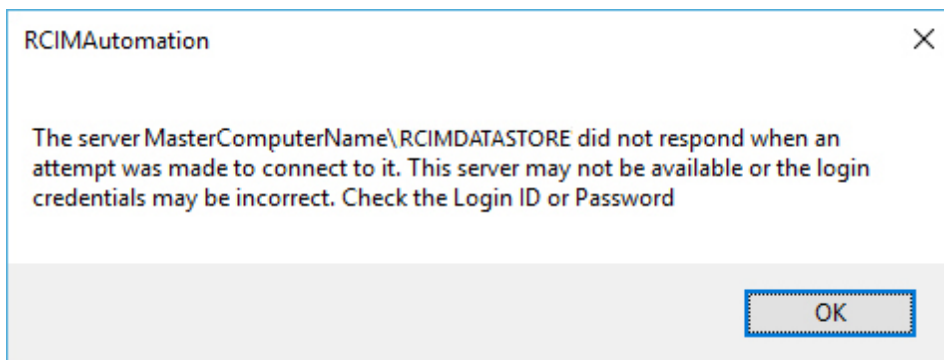
Once the SQL Server has been selected via the drop-down option or manually entered, select the 'Verify Server' button on the left side of the Database Manager:



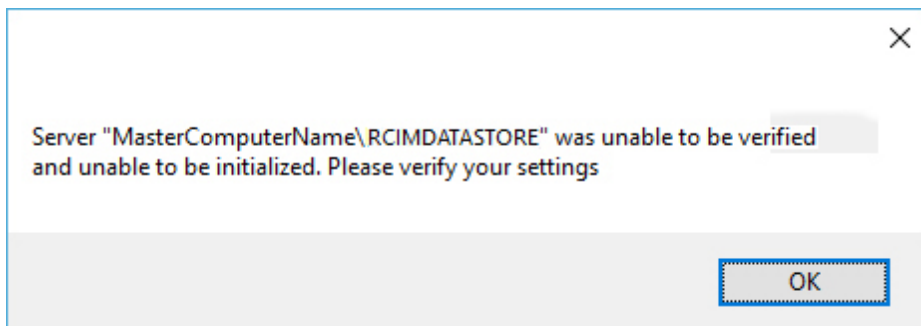
If the SQL Server was found, you will receive this message:



If the SQL Server was not found, you will receive this message:



Followed by:



At this point, verify your connection name and try again. Once you have selected an available server, see ['Set as Current'](#) section for more information, and connected successfully to it you can close the Database Manager.

Note: Once a Shared Database is setup, you may want to select the option '[Utilize Manager Record Locking \(Advanced Database Sharing\)](#)' under the 'Advanced Settings' section.

Installing on a Stand-Alone Server

Router-CIM Automation Suite uses SQL Express as its database system.

In order to setup a shared database using a stand-alone server, SQL Express needs to be installed on the stand-alone server following these instructions.

Select the SQL Express RCIM Installer download through the link below:

[SQL Express RCIM Installer](#)

Once the installer has downloaded, folder location is not important, run the SQLInstallerFullBuild.exe file. Follow the prompts to install the SQL Express on the stand-alone server.

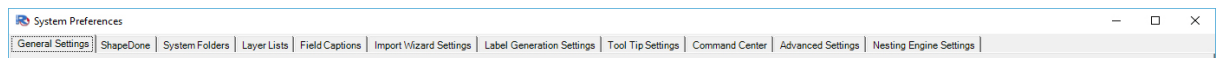
Once the installation has been completed, you will have the DB Manager icon on your servers desktop.



For more information on this, refer to the ['Database Manager'](#) section.

You can now continue setting up the client computers by following the ['Setting up a Shared Database'](#) section.

Settings



There are several settings that affect the entire system. These settings are broken down into the following categories:

[General Settings](#)

[ShapeDone Settings](#)

[System Folder Settings](#)

[Layer List Settings](#)

[Field Caption Settings](#)

[Import Wizard Settings](#)

[Label Generation Settings](#)

[Tool Tip Settings](#)

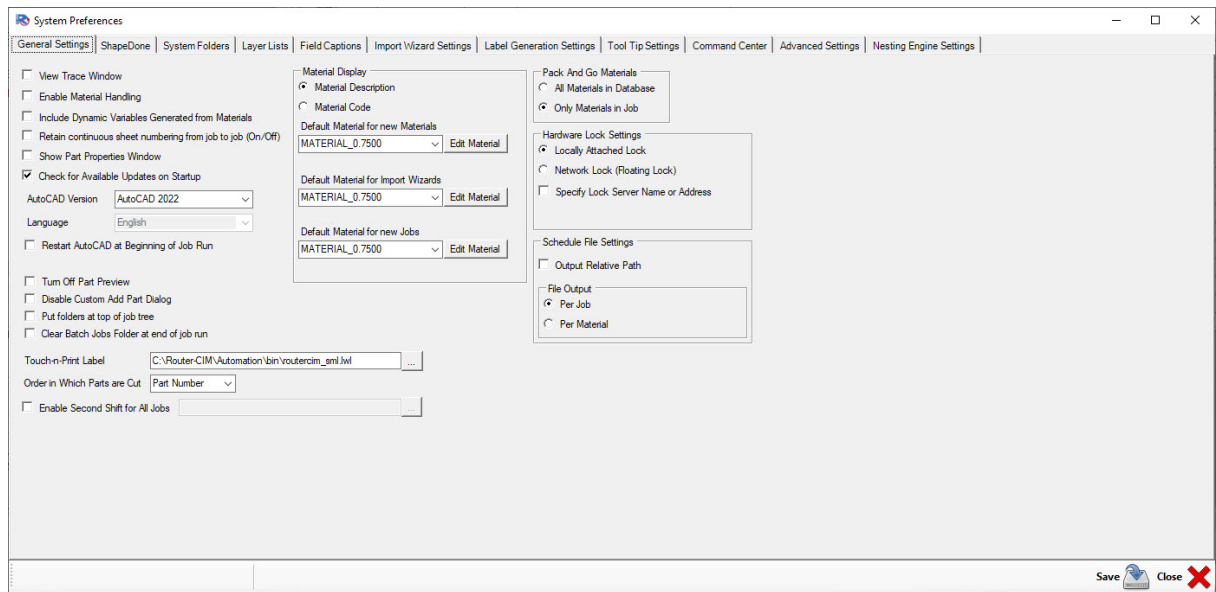
[Command Center](#)

[Advanced Settings](#)

[Nesting Engine Settings](#)

General Settings

The General Settings are miscellaneous settings that affect how Router-CIM Automation Suite runs and looks, and do not fit into the other categories of settings. Each one will be documented, and you can click on the area of the picture to be taken to that section's description.



View Trace Window

Checking this option will turn on an additional section to the status window that runs when processing a job. This option is only for debugging procedures and will only be used when directed by CIM-Tech.

Enable Material Handling

Turning this option on will allow the creation of code to handle the Material Handling Systems if your machine is equipped with one. See your post processor application notes for more information regarding the specific code for your material handling system.

Include Dynamic Variables Generated from Materials

Checking this option will force a dynamic variable to be created for each material with the material thickness as the value in the dynamic variable.

Retain Continuous Sheet Numbering from Job to Job (On/Off)

This option will set the job id in the NC code files to be continuous from job to job. The numbers start at 1 and run to 9999 and then will start over. You can force them to restart by un-checking this box and then running a job. Checking this box will force a continuous count to occur on each NC program until it reaches the top number (9999). Otherwise each job run starts with the first sheet as 001 and the second sheet as 002, etc.

Continuous Job ID setting requires:

- 1) Labels to be ON under File > Settings
- 2) Display repeated layouts OFF on Nesting Tab of job
- 3) Reset sheet counter per material OFF on Printing and Labels tab of job

Show Part Properties

Checking this box will make Router-CIM Automation Suite show the part properties window whenever you add a new part to a job (this was the version 2007 and older behavior). The default behavior is to add the part without showing the properties window.

Check for Available Updates

This option allows you to configure how the software keeps up-to-date. The Router-CIM Automation Suite software has the capability to check for available updates automatically to ensure you are always running the latest version of the software.

Note: Must have network connection

AutoCAD Version

This setting will have an option to change the AutoCAD version for Router-CIM Automation Suite if you have more than one version of AutoCAD installed on the computer that Router-CIM Automation Suite is running on. For currently supported AutoCAD versions, please review the ['System Requirements'](#) section.

Automation will set this for you if only 1 version is installed on your system, and no other choices will be available.

Language

Selecting this pull down list will change the Router-CIM Automation Suite interface to one of the supported languages, currently that is English. Other languages are pending and will be installed in later updates.

Restart AutoCAD at Beginning of Job Run

Turning on this option will start a new instance of AutoCAD when 'Run Job' is selected.

Turn off Part Preview

Turning on this option will disable the part preview window in Router-CIM Automation Suite so no part preview will be shown when a part is selected. This could allow some slower systems or systems with unsupported graphics cards to move through parts faster.

Disable Custom Add Part Dialog

Turning on this option will disable the custom add part dialog box when selecting the 'Add Parts' button. Use this feature for less graphics dependent computer systems.








Put Folders at the top of the Job Tree

Turning on this option will place the folders shown in the 'Job Tree' at the top of the column when sorted.

☐ Put folders at top of job tree

Name	Created On	Modified On
Job Library		
BOATRV_1_CODE_SINGLE_PART	7/28/2003 12:00:00 AM	9/4/2018 3:02:03 PM
CABINETS_1_CODE_SINGLE_PART	7/28/2003 12:00:00 AM	9/4/2018 3:02:07 PM
CABINETS_PRENEST_CODE_SINGLE_...	9/9/2016 12:00:00 AM	9/4/2018 3:02:00 PM
ADVANCED_NESTING_JOBS	9/8/2016 12:00:00 AM	
BASIC_NESTING_JOBS	9/8/2016 12:00:00 AM	
Batch Jobs		

☒ Put folders at top of job tree

Name	Created On	Modified On
 Job Library		
 ADVANCED_NESTING_JOBS	9/8/2016 12:00:00 AM	
 BASIC_NESTING_JOBS	9/8/2016 12:00:00 AM	
 BOATRV_1_CODE_SINGLE_PART	7/28/2003 12:00:00 AM	9/4/2018 3:02:03 PM
 CABINETS_1_CODE_SINGLE_PART	7/28/2003 12:00:00 AM	9/4/2018 3:02:07 PM
 CABINETS_PRENEST_CODE_SINGLE_...	9/9/2016 12:00:00 AM	9/4/2018 3:02:00 PM
 Batch Jobs		

Clear Batch Job Folder at end of job run

Turning on this option will delete the jobs listed in the Batch Jobs folder when the batch has been completed.

Touch and Print Label

This field shows the path and name of the Touch and Print Label file.

Order in Which Parts are Cut

When processing a job, you can select the order in which Router-CIM Automation Suite processes the parts.

- 1) Part Number: This will process the parts according to the Router-CIM part number assigned with the lowest number being processed first
- 2) Material: This will process the parts according to the material the part is assigned. The material order is alphabetical. This setting is ideal for multi-thread nesting or Cloud Nesting (Advanced Nesting Module).













Enable Second Shift Features for All Jobs

This is an advanced option available when you have a special program enabled called Second Shift. This option will turn on those special features for any job created or run. For more information contact CIM-Tech.













Material Display

The material display options of Material Code or Material Description relate to the way the materials are shown in the part window of a job.

Setting the option to Material Description will show the material as the text description listed for the material:

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
	Part #	PartName	Qty	Material				
	1	Sample1.dwg	4	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample1.dwg		
	2	Sample2.dwg	1	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample2.dwg		
	3	Sample3.dwg	2	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample3.dwg		
	4	Sample4.dwg	2	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample4.dwg		
	5	Sample5.dwg	4	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample5.dwg		
	6	Sample6.dwg	1	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample6.dwg		
	7	Sample7.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample7.dwg		
	8	Sample8.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample8.dwg		
	9	Sample9.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample9.dwg		
	10	Sample10.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample10.dwg		
	11	Sample11.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample11.dwg		
	12	Sample12.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample12.dwg		

Where setting the option to Material Code will show that same material listed by the material code which is a maximum of 8 characters:

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
	Part #	PartName	Qty	Material				
	1	Sample1.dwg	4	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample1.dwg		
	2	Sample2.dwg	1	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample2.dwg		
	3	Sample3.dwg	2	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample3.dwg		
	4	Sample4.dwg	2	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample4.dwg		
	5	Sample5.dwg	4	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample5.dwg		
	6	Sample6.dwg	1	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample6.dwg		
	7	Sample7.dwg	3	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample7.dwg		
	8	Sample8.dwg	3	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample8.dwg		
	9	Sample9.dwg	3	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample9.dwg		
	10	Sample10.dwg	3	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample10.dwg		
	11	Sample11.dwg	3	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample11.dwg		
	12	Sample12.dwg	3	MAT_500		C:\rcim_work\Automation\Samples\DWGS\Sample12.dwg		

Default Material for New Materials

The material selected in this field will be used as the default parameters for when creating a new material.

Default Material for Import Wizards

The material selected in this field will be used as the default material when using the import wizards and no material is defined.

Default Material for New Jobs

The material selected in this field will be used as the default material when "New Job" is selected.

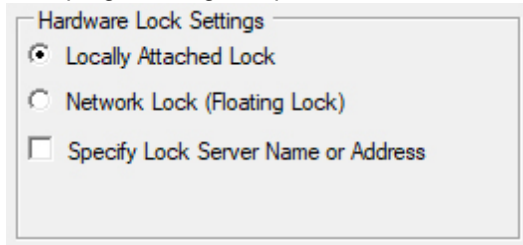
Pack and Go Materials

When making a Pack and Go file, you can have either ALL the materials in the whole database packaged into the Pack and Go file, or just the materials that relate to the job you are packing.

Hardware Lock Settings

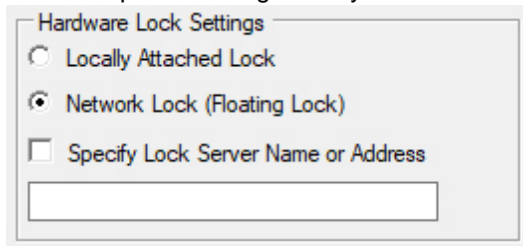
If this option is checked, the Router-CIM Automation Suite hardware lock will need to be located on the computer that is running Router-CIM Automation Suite. If this option is not checked, the hardware lock could possibly be located on another computer, and if it is on the same TCP/IP network then Router-CIM Automation Suite should find it and use it, provided it is not in use by another computer running Router-CIM Automation Suite. Installing the lock on the computer running Router-CIM Automation Suite is by far the most reliable means of insuring that no hardware lock issues occur.

1) Locally Attached Lock: The USB hardware lock for Router-CIM would need to be connected to the local programming computer



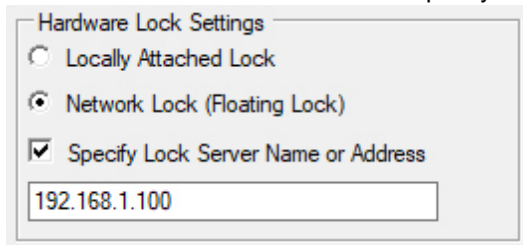
The screenshot shows a dialog box titled "Hardware Lock Settings". It contains three options: "Locally Attached Lock" (selected with a radio button), "Network Lock (Floating Lock)" (unselected), and "Specify Lock Server Name or Address" (unselected checkbox). The "Locally Attached Lock" option is highlighted with a blue selection bar.

2) Network Lock (Floating Lock): The USB hardware lock for Router-CIM can be connected to a server or other computer as long as they are on the same network.



The screenshot shows the "Hardware Lock Settings" dialog box. The "Network Lock (Floating Lock)" option is selected with a radio button. The "Specify Lock Server Name or Address" checkbox is unselected, and there is an empty text input field below it.

3) Specify Lock Server Name or Address: The USB hardware lock for Router-CIM can be connected to a server or other computer but if they are on different networks/subnets, you would need to select 'Specify Lock Server Name or Address' and specify the Server Name or IP Address.

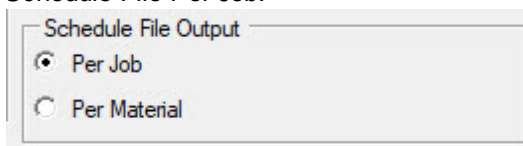


The screenshot shows the "Hardware Lock Settings" dialog box. The "Network Lock (Floating Lock)" option is selected with a radio button. The "Specify Lock Server Name or Address" checkbox is checked. The text input field below it contains the IP address "192.168.1.100".

Schedule File Output

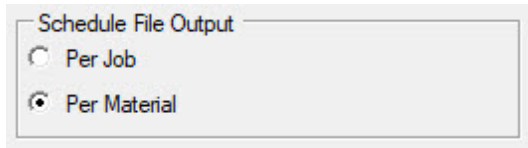
If a Schedule file is compatible with your CNC machine, Router-CIM Automation Suite can create the schedule file either by Job (all NC code files in one schedule file) or it will create separate schedule files based on the amount of different materials in the job run.

Schedule File Per Job:



The screenshot shows a dialog box titled "Schedule File Output". It contains two options: "Per Job" (selected with a radio button) and "Per Material" (unselected). The "Per Job" option is highlighted with a blue selection bar.

Schedule File Per Material:

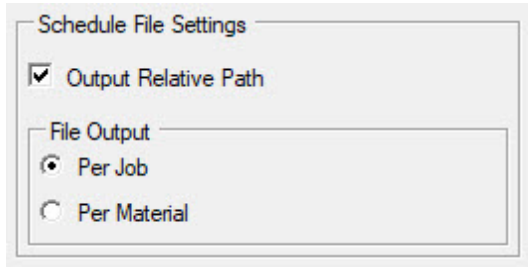


Schedule File Output

☐ Per Job

☒ Per Material

If you want the schedule file to also include the full output file path location for the NC Code files, select the 'Output Relative Path' option.



Schedule File Settings

☒ Output Relative Path

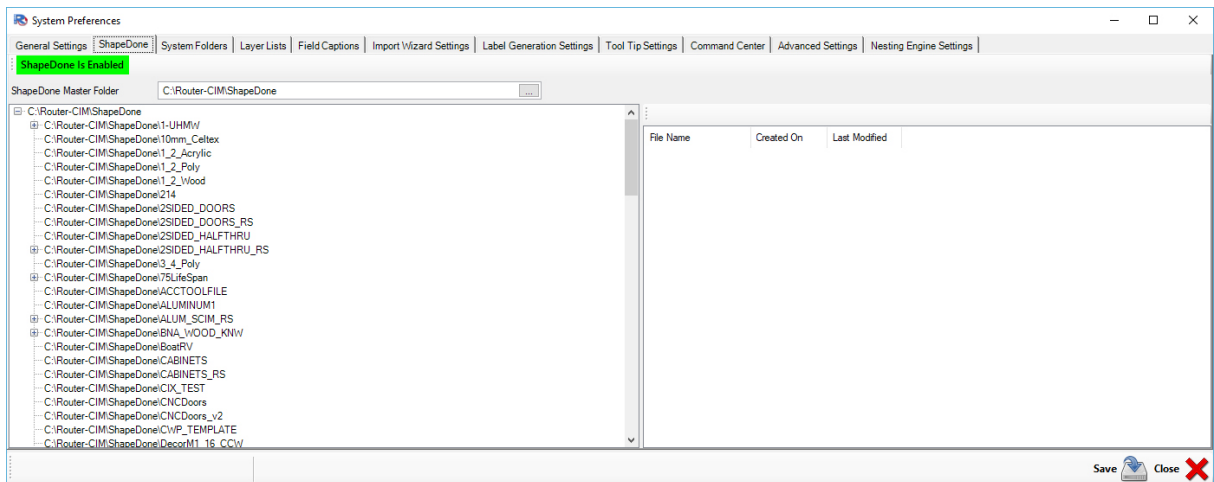
File Output

☒ Per Job

☐ Per Material

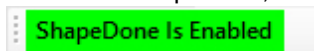
ShapeDone Settings

Enabling Shape Done will allow any parts that are used in a job, and have not been altered since they were last run to pass straight to nesting without having to be re-cut during the job run. This saves considerable time if you have jobs that re-use parts.

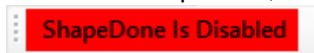


Enable ShapeDone

To Enable ShapeDone, select this button so it looks like below:



To Disable ShapeDone, select this button so it looks like below:



ShapeDone needs to be enabled in three locations:

- 1) File > Settings, under the 'ShapeDone' tab
- 2) On the [Job Settings](#) tab of the currently selected job, check the Enable ShapeDone box.
- 3) On the properties for the part 'Use ShapeDone Part When Possible' must be checked.

The first is a onetime setting. It turns ShapeDone on/off for the entire system.

The second allows you to turn ShapeDone on/off per job. You can default this to enabled by enabling it on a job and going to Tools > [Save Current Settings as Job Defaults](#).

The third allows ShapeDone to be used per part. Also on the [Job Settings](#) tab is a 'Use ShapeDone' option at the bottom under 'New Part Defaults'. This will automatically check the 'Use ShapeDone Part When Possible' box when adding any new parts to a job.

NOTE: ShapeDone cannot be used when 'Code as Single Part' or 'Nest and Code as Single Part' are selected under the [Part Properties](#) section.

ShapeDone Master Folder

Display and setting of the ShapeDone folder path and name.

ShapeDone Folders

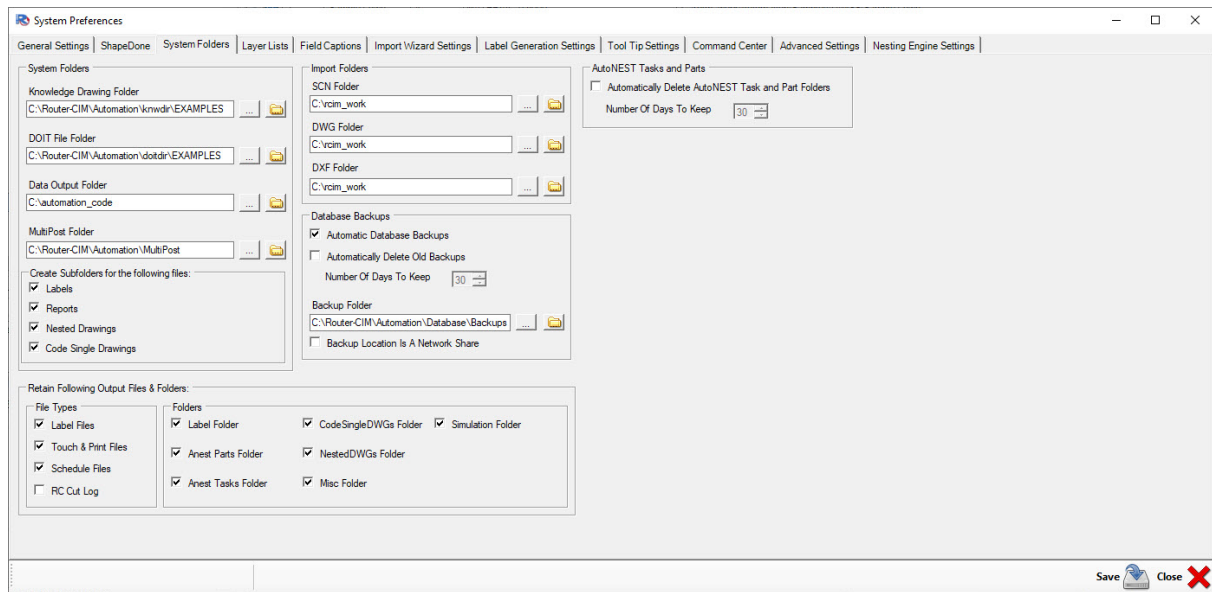
This side of the window displays the ShapeDone folders. One folder is created automatically for each knowledge drawing in the Router-CIM Automation Suite knowledge drawing folder.

ShapeDone Files

This side of the window displays the ShapeDone files in each of the folders. If you select a folder on the left, the files in that folder will show on the right.

System Folder Settings

The system folders settings are where you can change the path and folder locations for Router-CIM Automation Suite to find certain files for use.



System Folders

Knowledge Drawing Folder:

This is the path to the folder where you store your knowledge drawings.

By default it is C:\Router-CIM\Automation\knwdir\EXAMPLES.

DOIT File Folder:

This is the path to the folder where you store your DOIT files.

By default it is C:\Router-CIM\Automation\doitdir\EXAMPLES.

Data Output Folder:

This is the path to the folder where all the job files and NC code will be saved when a job has run.

By default this is C:\automation_code.

MultiPost Folder:

This is the path to the folder where all the job files and NC code will be saved when a job has run.

By default this is C:\Router-CIM\Automation\PackAndGo\EXAMPLES.

Please refer to the Multi Post section for more information on how to setup up a Multi Post configuration.

Import Folders

SCN Folder:

This is the path to the folder where Router-CIM Automation Suite will look for macros when importing them into a job if the path is not specified.

By default the path is set to C:\rcim_work

DWG Folder:

This is the path to the folder where Router-CIM Automation Suite will look for DWG files when importing them into a job if the path is not specified.

By default the path is set to C:\rcim_work

DXF Folder:

This is the path to the folder where Router-CIM Automation Suite will look for DXF files when importing them into a job if the path is not specified.

By default the path is set to C:\rcim_work

Create Subfolders for the following files:**Labels:**

Checking this box will create a separate folder in the result folder for all the label related files.

Reports:

Checking this box will create a separate folder in the result folder for all the report data related files.

Nested Drawings:

Checking this box will create a separate folder in the result folder for all the nested drawings.

Code Single Drawings:

Checking this box will create a separate folder in the result folder for the drawings of each part that was checked as code single part.

Database Backups**Automatic Database Backups**

Checking this box will force Router-CIM Automation Suite to back up the databases when Router-CIM Automation Suite is loaded for the first time each day. The databases will be stored in a folder on your C: drive for later restoration. The folder is C:\Router-CIM\Automation\Database\Backups or you can set the folder that you would like the backups to go to. Inside that folder will be a other folders with the date and time stamp as a name. These contain the databases for your system that were present on that date.

This backup happens once each day, on the first start of Router-CIM Automation Suite, so the database is backed up in the state it was in when last used.

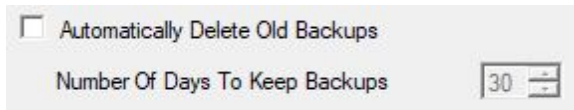
These back up files can be large and if you do not clean out the Backups folder, you can have many, many large files in there. This could take considerable drive space. These files could be copied onto a CD or some other permanent storage frequently and this folder cleaned of old, out of date files.

Removing the check from this box will force Router-CIM Automation Suite to ignore the back up procedure. This is dangerous because if you make a mistake in your settings or jobs, there is no backup copy and no way to retrieve lost data or settings. CIM-Tech cannot get data back from the database once it is changed. The backup is your best solution to retrieving lost settings or materials, etc.

Automatically Delete Old Backups

These back up files can be large and if you do not clean out the Backups folder, you can have many, many large files in there. This could take considerable drive space. These files could be copied onto a CD or some other permanent storage frequently and this folder cleaned of old, out of date files.

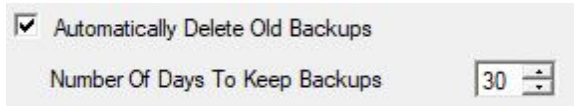
If Automatic Database Backups is enabled, this option will continue to create backups and keep the files in the defined location



☐ Automatically Delete Old Backups

Number Of Days To Keep Backups

If this option is enabled, you will be able to define the amount of days to keep the database backup files. Use the up and down arrow to adjust the amount of days that you would like to keep the database backup files for.

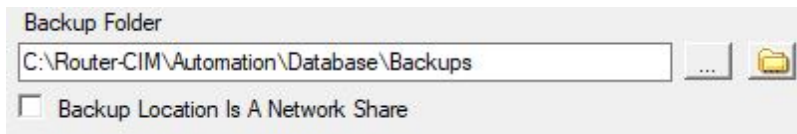


☒ Automatically Delete Old Backups

Number Of Days To Keep Backups

Backup Folder

This is the path to the folder where you store your 'Automatic Database Backups'. The folder, by default, is C:\Router-CIM\Automation\Database\Backups. You can set this to a shared drive location by identifying the file path you would like.



Backup Folder

☐ Backup Location Is A Network Share

The 'Backup Location Is A Network Share' option will automatically be checked by Router-CIM if it sees that the location is a shared drive.

Note: Shared folders must have a mapped drive letter.

Please refer to the ['Restore New'](#) or ['Restore Over'](#) section for information regarding bringing in a backup database.

Retain Following Output Files & Folders:

File Types:

- Label Files** - Checking this box will retain the created label files.
- Touch and Print Files** - Checking this box will retain the created Touch and Print files.
- Schedule Files** - Checking this box will retain the created schedule file.
- RC Cut Log** - Checking this box will retain the created RC_CutLog.log file.

Folders:

- Label Folder** - Checking this box will retain the created label folder (if selected under 'Create subfolders for the following files').
- Anest Parts Folder** - Checking this box will retain the created Anest Parts folder.
- Anest Tasks Folder** - Checking this box will retain the created Anest Tasks folder.
- CodeSingleDWGs Folder** - Checking this box will retain the created CodeSingleDWGs folder (if selected under 'Create subfolders for the following files').
- NestedDWGs Folder** - Checking this box will retain the created NestedDWGs folder (if selected under 'Create subfolders for the following files').
- Misc Folder** - Checking this box will retain the created Misc folder.
- Simulation Folder** - Checking this box will retain the created Simulation folder if Cut-SIM has been installed
- WMF Folder** - Checking this box will retain the created WMF image folder.

AutoNEST Task and Parts

Automatically Delete AutoNEST Tasks and Parts Folder

Checking this box will force Router-CIM Automation Suite to delete any files that are within the AutoNEST Tasks and Parts folders.

If this option is enabled, you will be able to define the amount of days to keep AutoNEST tasks and parts. Use the up and down arrow to adjust the amount of days that you would like to keep the AutoNEST files for.

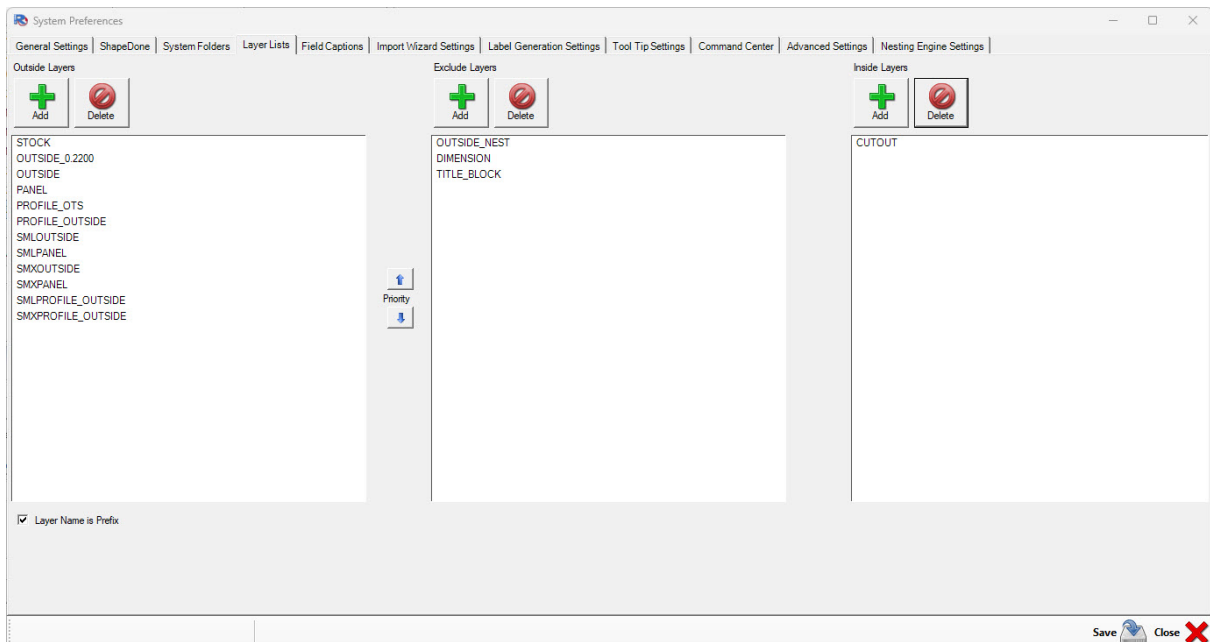
☒ Automatically Delete Old Backups

Number Of Days To Keep Backups

Layer List Settings

The Outside Layer lists are used by Router-CIM Automation Suite to associate which layers will contain the outside geometry used for nesting and which layers to exclude from a drawing when considering a part for nesting.

Note: If you are receiving 'Part not OK for Nesting' or in your nested drawings you are seeing a part '1X1', you have not properly entered in the correct Outside layer into the 'Outside Layers' column.



Outside Layers

In this list you should include any layer that will contain geometry that will be the outside of the part when the part will be nested. Router-CIM Automation Suite will check each part and when the geometry of the outside of the part is found, it is checked against this list and then the part is passed to the nest program.

A common cause of parts not showing up in a nest is that the parts outside layer was not included in this list. Another common mistake is to have an inside cut be on a layer that is in this list also, that can cause parts nesting on the top of other parts.

An important note is that Router-CIM Automation Suite looks at this list in order until it finds a match, and so moving a layer higher in the priority in the list will cause Router-CIM Automation Suite to find one layer before another.

Using the 'Layer Name is Prefix' option will cause Router-CIM Automation Suite to look at the name given here as just the prefix of the whole layers name. For instance if you had geometry on both layer OutsideDoorBigTool and OutsideDoorSmallTool and put the layer name OutsideDoor in the list and checked 'Layer Name is prefix'...than both those layers would be considered as outside geometry.

Exclude Layers

Any layers listed in this window will be ignored by Router-CIM Automation Suite for the purpose of cutting and nesting. You can include dimension layers here for example as you might have them in a drawing but do not want to cut or nest them.

Inside Layers

In this list you should include any layer that will contain geometry that will be an inside thru feature on the part. Router-CIM Automation Suite will check each part and when the geometry of the inside feature of the part is found on the defined layers, it is checked against this list when using the ['Small Inside Feature Options'](#).

Field Caption Settings

The Field Captions window displays the label captions that are shown in the Part Properties window. For instance these settings:

System Preferences

General Settings | ShapeDone | System Folders | Layer Lists | **Field Captions** | Import Wizard Settings | Label Generation Settings | Tool Tip Settings | Command Center | Advanced Settings | Nesting Engine Settings

Specify the label captions that you would like to use on the Part Properties Window

Record Entry	Part Name
Record Material	Material
Record Qty	Quantity
Record X-Dim	Length
Record Y-Dim	Width
Record Z-Dim	Depth

Record Description	Record Desc
Label Field 1	Customer Info 1
Label Field 2	Customer Info 2
Label Field 3	Customer Info 3
Label Field 4	Customer Info 4
Label Field 5	Customer Info 5
Label Field 6	Customer Info 6
Label Field 7	Customer Info 7
Label Field 8	Customer Info 8

Material Attributes	
Attribute 1	Underscore
Attribute 2	Two
Attribute 3	Three
Attribute 4	Four
Attribute 5	Five
Attribute 6	Six
Attribute 7	Seven
Attribute 8	Eight
Attribute 9	Nine
Attribute 10	Ten

Save Close

These settings will display like this on the Part Properties window of any part in a job once they are set. Select a part and then either double click on it or select Part Properties to see the changes.

For instance, changing the Label Fields 1-8 shown above from 'Customer Info 1-8' to more specific data like this:

System Preferences

General Settings | ShapeDone | System Folders | Layer Lists | Field Captions | Import Wizard Settings | Label Generation Settings | Tool Tip Settings | Command Center | Advanced Settings | Nesting Engine Settings

Specify the label captions that you would like to use on the Part Properties Window

Record Entry	Part Name
Record Material	Material
Record Qty	Quantity
Record X-Dim	Length
Record Y-Dim	Width
Record Z-Dim	Depth
Record Description	Record Desc
Label Field 1	Customer Name
Label Field 2	Customer Address 1
Label Field 3	Customer Address 2
Label Field 4	Customer City
Label Field 5	Customer State
Label Field 6	Customer Zip
Label Field 7	Customer Phone
Label Field 8	Customer Email

Material Attributes

Attribute 1	Underscore	Attribute 6	Six
Attribute 2	Two	Attribute 7	Seven
Attribute 3	Three	Attribute 8	Eight
Attribute 4	Four	Attribute 9	Nine
Attribute 5	Five	Attribute 10	Ten

Save Close

Would show up in the Part Properties like this:

Part Properties

Part Information | Knowledge Information

Part Name: C:\vcim_work\Automation\Samples\DWGS\CABINETS1_1.dwg

Record Desc: CABINETS1_1

Material: MATERIAL_0.7500

Quantity: 18

Filler Qty: 0

Maximize parts on sheet of material

Label Information

Customer Name	
Customer Address 1	
Customer Address 2	
Customer City	
Customer State	
Customer Zip	
Customer Phone	
Customer Email	

Inherit Label Data from Job

Veneer Match Parameters

Name	
Location Point	
Rotation	

Part Options

Ignore Layer Panel

Start Point on longest side of feature

Manual Origin Part

Use ShapeDone Part when possible

Part Machining Operations

Nest and Code As Single Part

Nest Part

Code As Single Part

Has an Associated Backside Macro

Is an Irregular Stock Shape

Output To Saw

Macro Dimensions

Length	25.25
Width	2.75
Depth	0.75

Part Orientation

Rotate Part

Rotate Angle: 90

Mirror Part

Mirror Axis: Horizontal

Nest Rotation: Same as Material

Save Close

And in the 'Printing and Labels' tab like this:

The screenshot shows the 'Printing and Labels' tab with the 'Labeling' section expanded. The 'Labels' list contains the following fields, with a red box highlighting the first seven:

Label	Input Field
Customer Name	<input type="text"/>
Customer Address 1	<input type="text"/>
Customer Address 2	<input type="text"/>
Customer City	<input type="text"/>
Customer State	<input type="text"/>
Customer Zip	<input type="text"/>
Customer Phone	<input type="text"/>
Customer Email	<input type="text"/>

Material Attributes

Changing the fields for the 10 material attributes listed will cause these attributes to be listed in the material database as they are listed in this window. For instance if you were to change these options to:

The screenshot shows the 'Material Attributes' window with 10 attributes and their values:

Attribute	Value
Attribute 1	Underscore
Attribute 2	Two
Attribute 3	Three
Attribute 4	Four
Attribute 5	Five
Attribute 6	Six
Attribute 7	Seven
Attribute 8	Eight
Attribute 9	Nine
Attribute 10	Ten

Then click on OK and open up any material in the material database, you would see the material attributes listed the same.

Material Properties

Material Description: MATERIAL_0.7500

Material Code: MAT_750 Material Handling Code: ☐ Z0 is top of Material

Stock Settings

Sheet Stock Y Dim: 49.0000 Quantity: 999

Sheet Stock X Dim: 97.0000 Priority: 5

Thickness: 0.7500

Bridge Width: 0.6250

Cost: 0.00

Edge Allowances

Top: 0.1250

Left: 0.1250 Right: 0.1250

Bottom: 0.1250

Irregular Stock Edge Allowance: 0.1250

Material Rotation

☐ Allow Part Rotation (All) ☐ No Part Rotation (0)

☐ Part Rotation (0 90 180) ☐ Part Rotation (0 90)

☐ Grain Rotation (90 270) ☒ Grain Rotation (0 180)

☐ Custom Rotation:

Material Attributes

Underscore: Six:

Two: Seven:

Three: Eight:

Four: Nine:

Five: Ten:

Scrap Management | **Advanced Nesting Parameters**

☒ Make a Scrap Cut ☐ Inventory Scrap ☐ Use Scrap

☐ Inventory to other Material

Scrap

+ Add Scrap - Delete Scrap

ID	YDim	XDim	Qty	Priority	Cost	Bin	Availability

Scrap Cut Properties

☐ Longest Side Only

☐ Shortest Side Only

☒ Shortest Side First

☐ Longest Side First

☐ Vertical Only

☐ Horizontal Only

☐ Vertical First

☐ Horizontal First

☐ Combined

Min. X: 12.0000

Min. Y: 12.0000

Distance from Part in X: 0.3750

Distance from Part in Y: 0.3750

Scrap Cut Extension Distance: 0.3750

Save Close

Make Default Material For Import Wizards

Import Wizard Settings

System Preferences

General Settings | ShapeDone | System Folders | Layer Lists | Field Captions | **Import Wizard Settings** | Label Generation Settings | Tool Tip Settings | Command Center | Advanced Settings | Nesting Engine Settings

File Type: SCN

Right Click Excel Default Import Wizard Format: SOLID-CIM3D

Right Click CSV Default Import Wizard Format: SOLID-CIM3D

Select and Go Default Import Wizard Format: SOLID-CIM3D

Template Job: CABINETS_1

CSV Import Folder: C:\cim_work\

Default Assembly Folder:

Save Close

File Type

This will allow you to set SCN, DWG, or DXF files as the default file type for the import wizard to import if the files extensions are not provided in the import file.

Right Click Excel Default Import Wizard Format

If you saved an Import Wizard format it would show in this drop down. For example, If you import door_import.xls, then assign the columns, save the format and call it 'Solid-CIM3D', it will show in this list. Then you can select 'Solid-CIM3D' as the default wizard format for Excel (xls orxlsx files). Then if

you right-click an Excel file, and select cut with Router-CIM Automation Suite, 'Solid-CIM3D' will be used to format the columns on import.

Right Click CSV Default Import Wizard Format

If you saved an Import Wizard format it would show in this drop down. For example, If you import door_import.CSV, then assign the columns, save the format and call it 'Solid-CIM3D', it will show in this list. Then you can select 'Solid-CIM3D' as the default wizard format for Comma Delimited (CSV files). Then if you right-click a CSV file, and select cut with Router-CIM Automation Suite, 'Solid-CIM3D' will be used to format the columns on import.

Select and Go Default Import Wizard Format

If you saved an Import Wizard format it would show in this drop down. For example, If you import door_import.CSV, then assign the columns, save the format and call it 'Solid-CIM3D', it will show in this list. Then you can select 'Solid-CIM3D' as the default wizard format for Select and Go. When you use the Select and Go import wizard, 'Solid-CIM3D' will be used to format the columns on import of either a CSV or Excel file.

Template Job


This selection will allow you to have a preset job setup for running a Right Click import. Multiple template jobs can be made but only one can be set to the default.

CSV Import Folder

Allows you to predefine the folder location where Router-CIM Automation Suite will open by default when using the CSV Import Wizard.

Default Assembly Folder

The default assembly folder specifies where the main assemblies of jobs will be located if an assembly is imported via a spreadsheet. Since an assembly is a part record in an import, and it contains other jobs, Router-CIM Automation Suite must know where to go looking for those jobs when it imports the assembly. To import an assembly spreadsheet, the part name must be the assembly followed by the file extension of .MJF (macro job format) in order for it to recognize the assembly.

 System Preferences

General Settings	ShapeDone	System Folders	Layer Lists	Field Captions	Import Wizard Settings
File Type					DWG
Right Click Excel Default Import Wizard Format					DoveTail_Template_Import
Right Click CSV Default Import Wizard Format					DoveTail_Template_Import
Select and Go Default Import Wizard Format					DoveTail_Template_Import
Template Job					
CSV Import Folder					
Default Assembly Folder					MACRO_ASSEMBLIES

Label Generation Settings

These options are available as system settings and will affect each job. Setting this option to '**No Labels**' will suppress the generation of any label files by Router-CIM Automation Suite for any job run from that point. Setting the option to '**Comma Delimited Labels**' will generate label files for each job in several formats.

Selecting the option '**Place Preview on Labels**' will generate a small thumbnail image of the part and place it on the label. The preview that is generated for the labels will be the size set under the 'Image Size (Width and Height)' option in pixels and the amount of whitespace (border) used for a border according to the setting 'Percent of Total Image To Be Used for Whitespace Border'.

To setup an RCIM Labeling Configuration, go to the '[Automation Label Settings](#)' section.

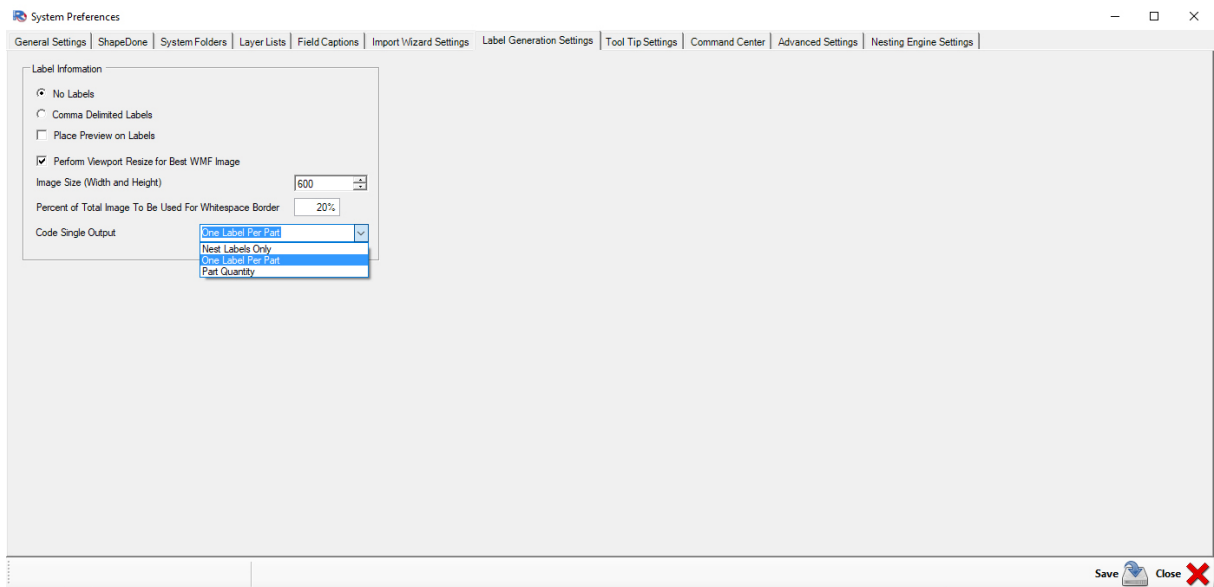
Code Single Output:

When running a Router-CIM Automation Suite job with the parts set to '**Nest and Code as Single Part**' or '**Code as Single Part**', you have the option to identify how you would like the labels for the single part files

Nest Labels Only - No labels will be produced for the single part files. If you have labeling enabled, you will only get the labels for the parts that were set to nest.

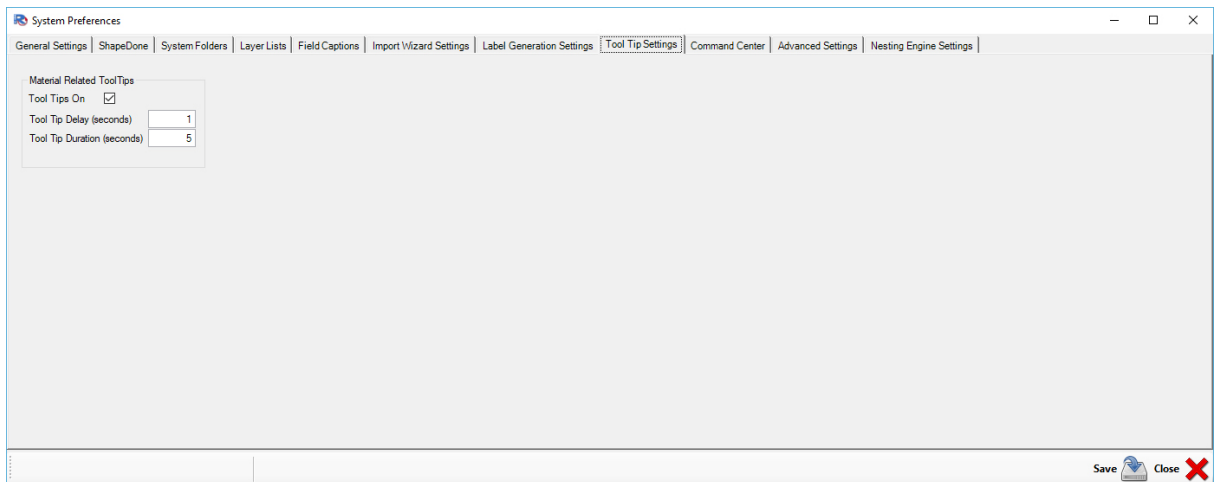
One Label Per Part - You will get 1 label per part for any parts that are set to '**Nest and Code as Single Part**' or '**Code as Single Part**'

Part Quantity - You will get the quantity of labels per part based on the job setup if the parts are set to '**Nest and Code as Single Part**' or '**Code as Single Part**'

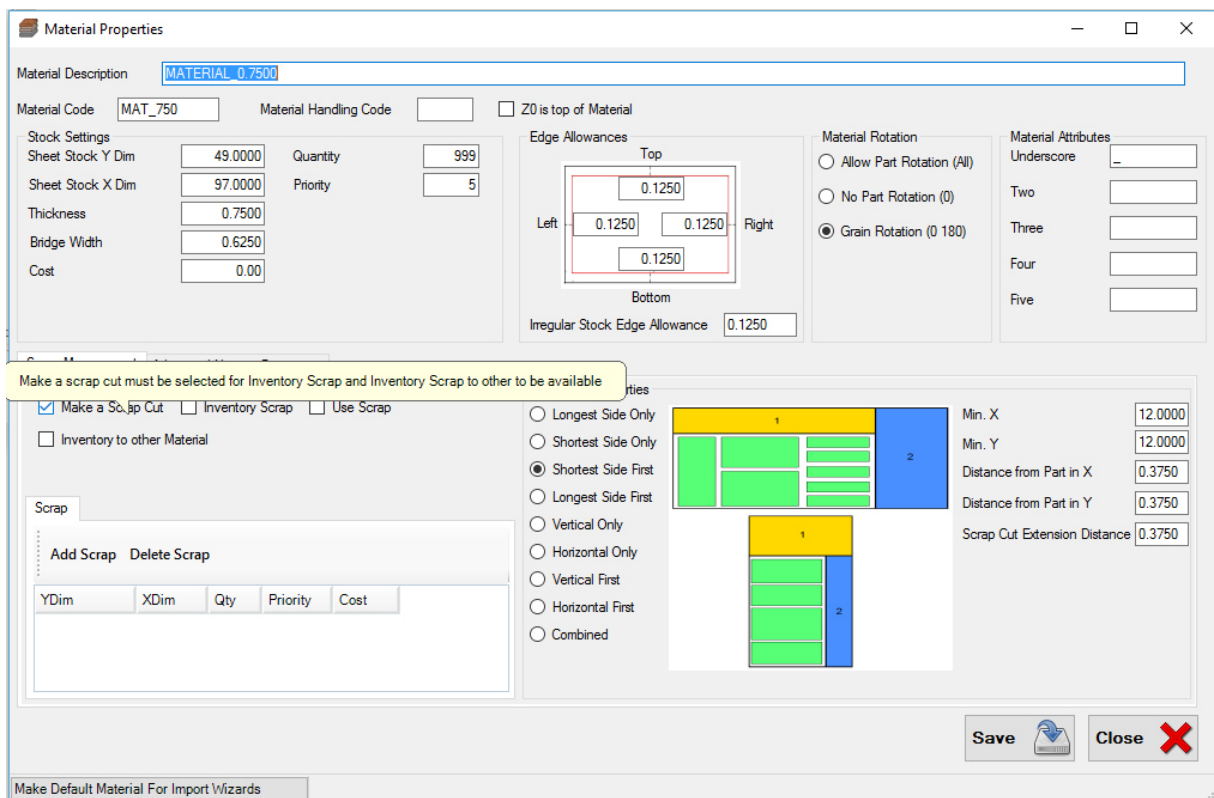


Tool Tip Settings

The 'Tool Tip Settings' control the informational 'Tips' that pop up when you hover over certain features. These tips give simple descriptions about what the feature is. These tips are available in the Materials database. You can see an example of one below.

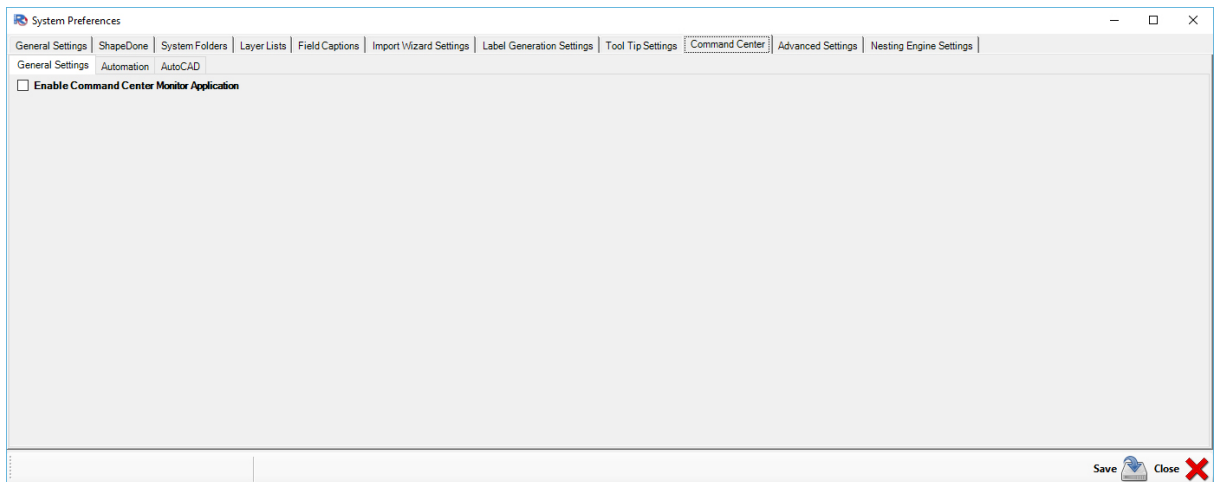


Here is an example of a Tool Tip:



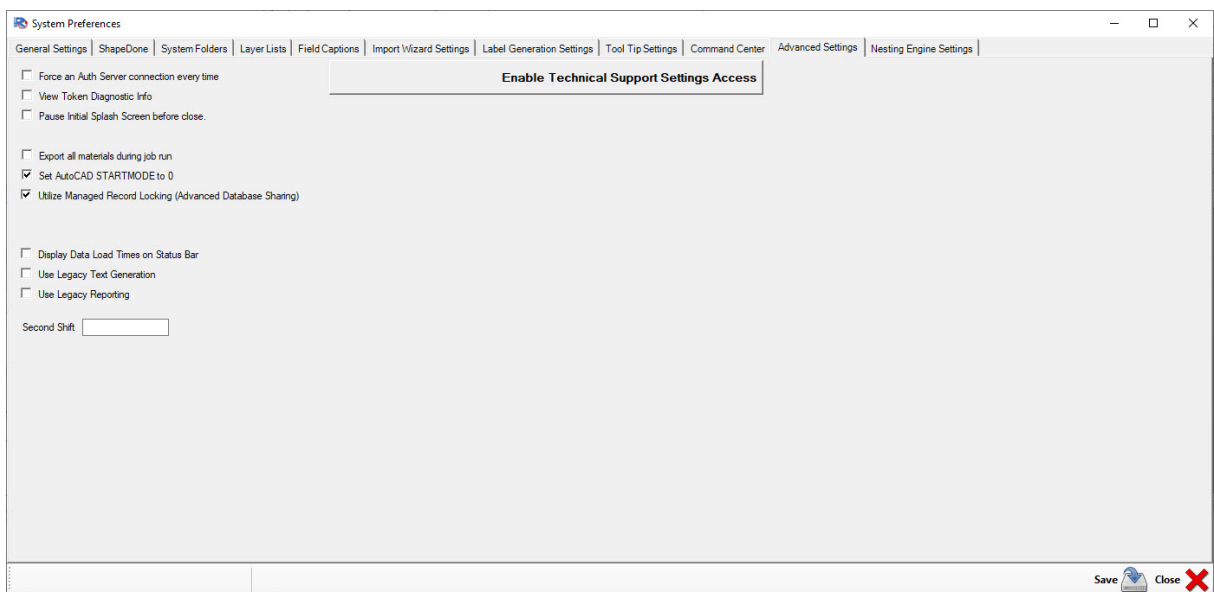
Command Center

The Command Center is for custom programming through CIM-Tech's AutoCIM program.



Advanced Settings

The Advanced Settings are primarily used for Technical Support and for Managed Record Locking when using [Shared Databases](#).



Export all materials during job run

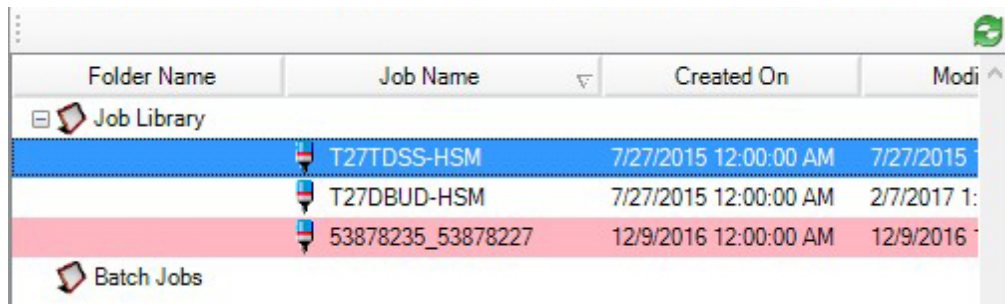
This option needs to be selected when using Multi-Stock material sizes and having unique materials created in the Material Database for each stock size available.

Utilize Managed Record Locking (Advanced Database Sharing)

Managed record locking for jobs prevents users in the shared database system from modifying a job that someone else is currently on. You can view data on the job, but any modifications made to the job are not saved when the focus moves to another job or Automation is closed. Access to part data is not enabled. Access to Dynamic variable information is read only.

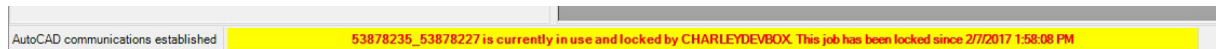
☒ Utilize Managed Record Locking (Advanced Database Sharing)

The Router-CIM Automation Suite interface will indicate what jobs are currently locked in the current database.



Folder Name	Job Name	Created On	Modified
Job Library			
	T27TDSS-HSM	7/27/2015 12:00:00 AM	7/27/2015
	T27DBUD-HSM	7/27/2015 12:00:00 AM	2/7/2017 1:
	53878235_53878227	12/9/2016 12:00:00 AM	12/9/2016
Batch Jobs			

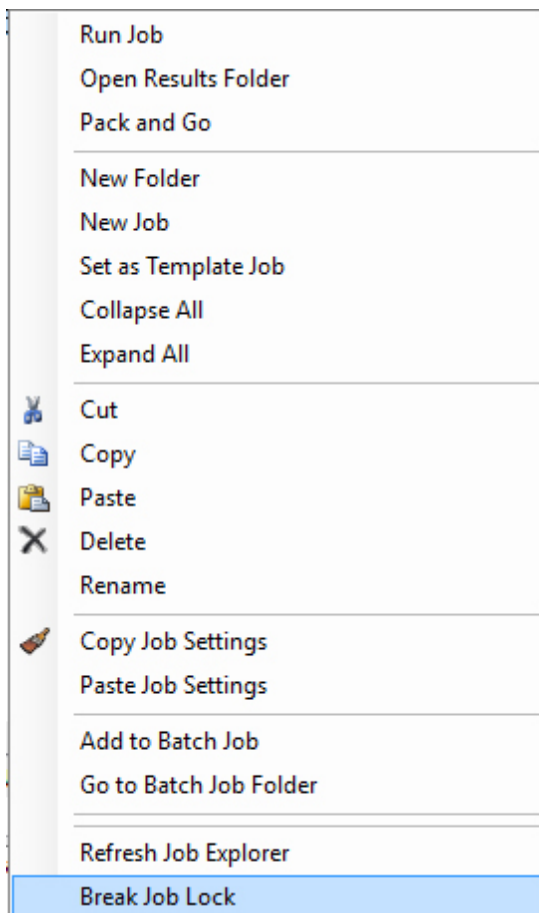
You will also see that if you select on the job that is in use, a message will appear at the bottom of the interface notifying you that it is in use and show you who is currently in the job:



AutoCAD communications established	53878235_53878227 is currently in use and locked by CHARLEYDEVBOX. This job has been locked since 2/7/2017 1:58:08 PM
------------------------------------	---

Breaking a Job Lock

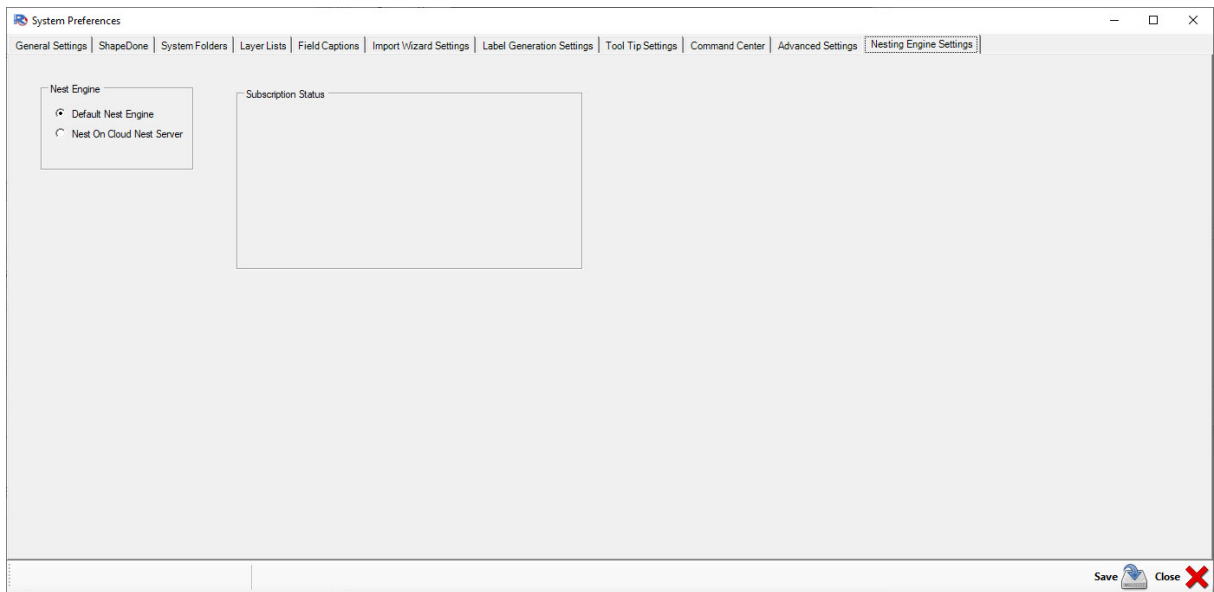
Sometimes it is necessary to “Break” a job lock. For instance, if one user leaves for the day and did not shutdown Router-CIM Automation Suite, leaving a job open, then that job is basically inaccessible to everyone else using this database until they return. In cases like this, you will want to free that job so someone else can work with it. To do this, right click the job that is locked, it will be red and select 'Break Job Lock'.



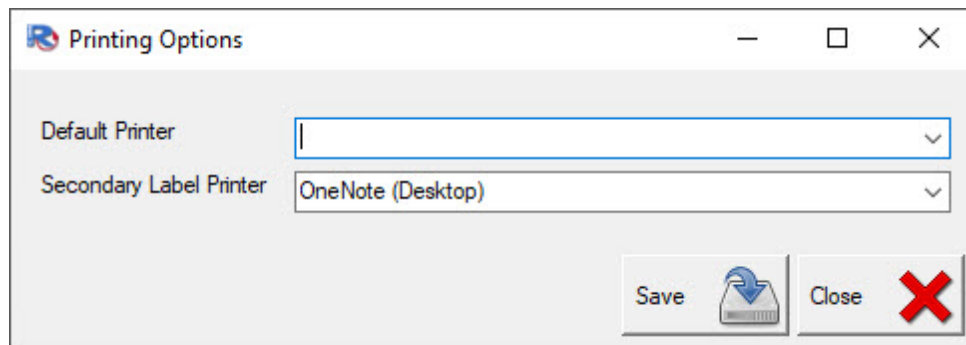
Nesting Engine Settings

The Nesting Engine Settings allow you to switch from the Default Nest Engine (local computer) to either Nest on Cloud Nest Server (cloud based nesting) or Just In Time Nesting (local computer with Cloud Nest capability).

Nest on Cloud Nest Server requires a Cloud Nest subscription. Please contact CIM-Tech for further information.



Print Options



Print Options will allow you to select the default printer for Router-CIM Automation Suite to print the nested sheets or single part prints during a job run. Any printer available to Windows in the Print Manager will be available in the list of printers selected here.

You can also select a second printer for using when 'Print Labels' is selected for your label configuration.

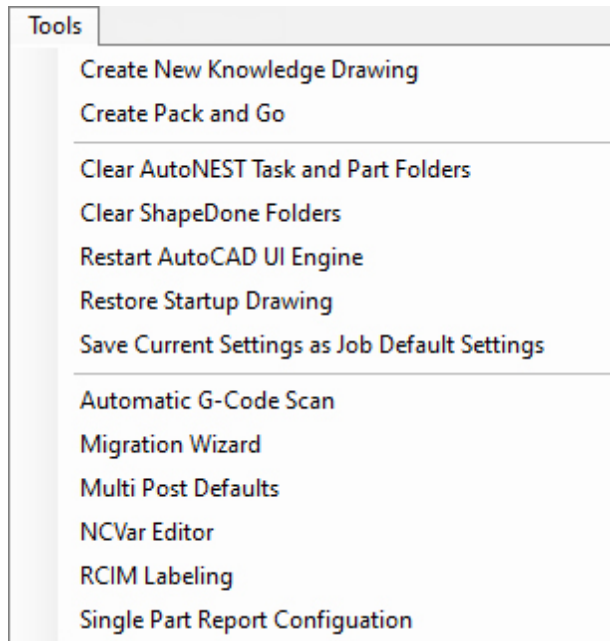
Exit

Choosing Exit, or using the 'X' in the upper right corner of the window will close the Router-CIM Automation Suite program, but before doing that a few internal functions are performed, such as saving all jobs and job related information, saving the screen size and position data, part preview window information, and saving the system defaults for use the next time Router-CIM Automation Suite is started.

Tools Menu

Router-CIM Automation Suite gives you tools menu to use for the most common items that are used.

Select a topic on the picture for more in-depth information.



Create New Knowledge Drawing

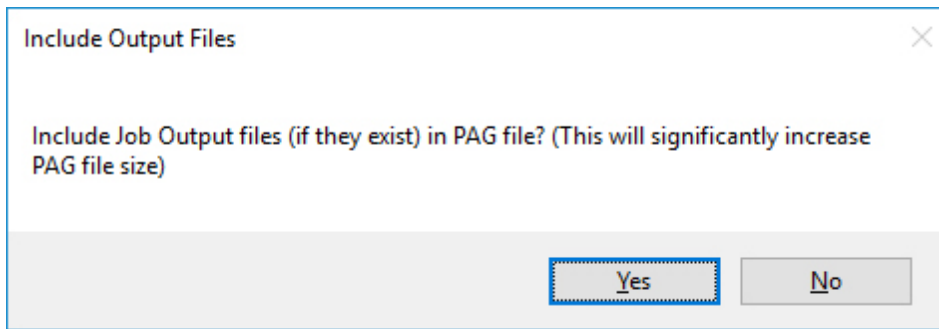
A new knowledge drawing can be created with this option, even if you do not currently have a job selected. Once this option is selected, you will get the Router-CIM Configuration Wizard where you select your machine upon AutoCAD opening. Router-CIM will then start, and you can import or create new knowledges and save the drawing in the correct location based on the ['System Folders'](#) settings.

You must use the SaveAs option and give the drawing a different name. The system opens up a drawing called NewKnow.DWG by default and this is write protected (set as Read-Only), so that you must use the SaveAs option and give your knowledge drawing a different name.

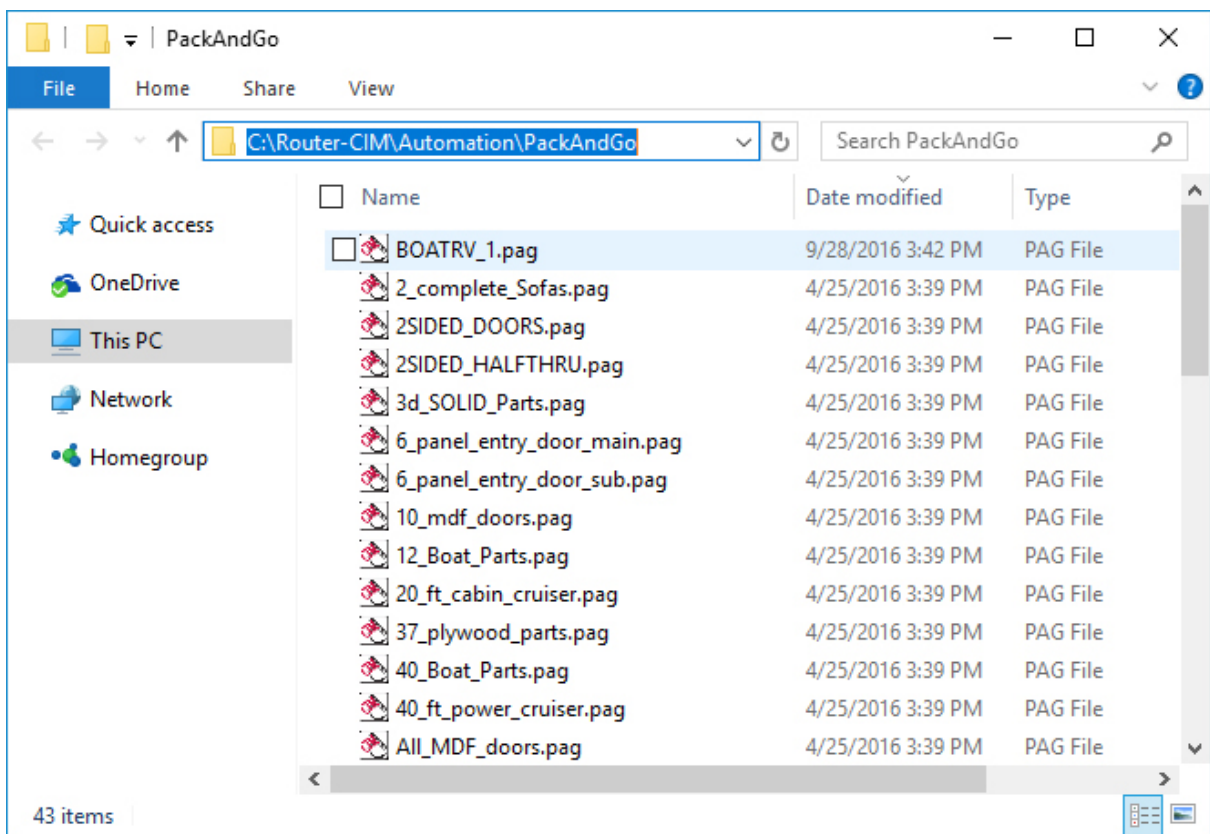
Pack and Go

The Pack and Go function will allow you to package any job or multiple jobs into just one file that can be sent to another user of Router-CIM Automation Suite who can then unpackage that job and have the same settings as you did on your Router-CIM Automation Suite system when the job was run.

The Pack and Go function will ask you if you want to place all the output files into the package (if any were made) so that any results from the job can be viewed by the other user.



Once you acknowledge this window, file explorer will open and you can then see the pack and go file, which can be attached to an email or transferred via some other means to the other user.



The folder, by default is C:\Router-CIM\Automation\PackAndGo.

Use Pack and Go as a Backup

Another use for the Pack and Go system is to create backups of jobs that have run, so that all the settings are stored in one location, regardless of the database. The pack and go function will place the following items into the PAG file.

- Any drawings, macros, or DXF files from the job, along with their path and folder if necessary.
- The Knowledge drawing used for the job.
- The DOIT file used for the job.

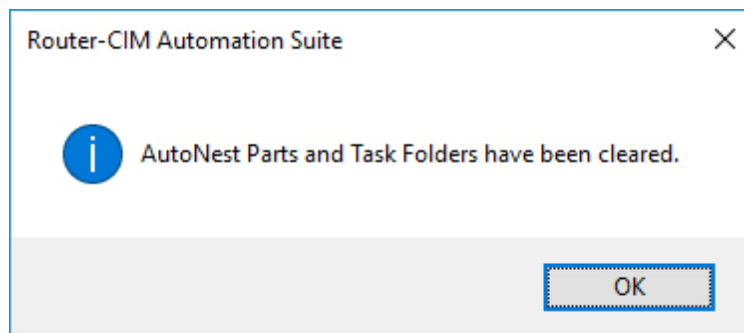
- All post processor files used for the job.
- All job related settings.
- The material used on the job.
- If specified, the entire results folder and contents.

Clear AutoNest Task and Part Folders

This option will remove all the contents of the AutoNEST parts folder and the AutoNEST tasks folder. After running many jobs, these folders can become very large and may contain previous copies of a job or the parts in a job that may not be current.

It is a good practice to clean these folders out from time to time.

Once you select this option, you will have a momentary pause and then the following confirmation is shown.



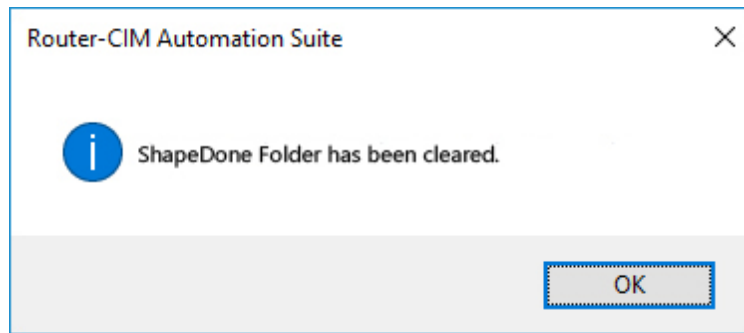
Clear ShapeDone Folder

This option will remove all the contents of the ShapeDone folder. After running many jobs, these folders can become very large and may contain previous copies of the parts in a job that may not be current.

It is a good practice to clean these folders out from time to time.

Note: Clearing the ShapeDone folder will reset your ShapeDone parts causing Router-CIM Automation Suite to reprocess the parts to refresh the ShapeDone folder.

Once you select this option, you will have a momentary pause and then the following confirmation is shown.



Restart AutoCAD UI (User Interface) Engine

The Restart AutoCAD UI (User Interface) Engine tool allows you to restart the Router-CIM Automation Suite AutoCAD session.

In some situations, the AutoCAD session that has been started when Router-CIM Automation Suite was opened may fail. The situation that arises is that a Router-CIM Error Report will be presented stating that there was an error writing to the pipe:

Router-CIM Error Report

Router-CIM Automation Suite has generated an error

Error Data

Information Title	Data
Begin Error Data	***** Begin Error Data *****
Primary Application Name	RCIMAutomation.clsWCFCCommunicationLibrary
Component Name	
Product Serial #	
File Name	
Function Name	GetPost
Exception Date/Time	9/29/2016 11:20:28 AM
Primary Error Message	There was an error writing to the pipe: The pipe is being closed. (232...
Inner Exception Message	The write operation failed, see inner exception. There was an error wr...
Machine Name	CIM-TECH
User Name	CIM-TECH\Sheldon
Monitor Size	
Boot Mode	
OS Version	Microsoft Windows NT 6.2.9200.0
Stack Application Name	RCIMAutomation
Error Stack Trace level 1	*****
Namespace	
Declaring Type Name	RealProxy
Name	HandleReturnMessage
CodeBase	
Native Offset	14493634
FileName	RCIMAutomation.exe
Line #	0
Column #	0

Return email address

Additional Notes

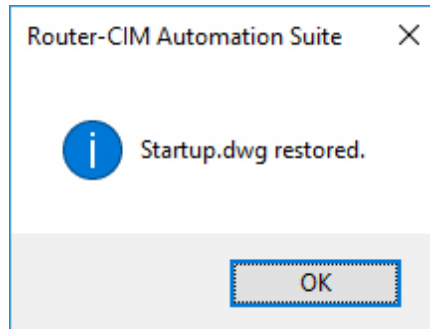
Transmit Error Data Copy error to clipboard OK

If this error presents itself, select the 'Restart AutoCAD UI (User Interface) Engine' option under the [Tools](#) drop down menu. This will restart the Router-CIM AutoCAD session for you.

Restore Startup Drawing

The startup drawing (Startup.dwg) contains various bits of data for Router-CIM Automation Suite to function. Occasionally that drawing can become corrupt due to an accidental save. If the drawing does become corrupt, this option will restore it to the default drawing status.

Once this option is selected, the following confirmation will appear.



Save Current Settings as Job Default Settings

Using this option will store the current settings in the currently selected job and make them the defaults for any new job that is created, or any new job that is made with the right click 'Cut with Router-CIM Automation Suite' option.

The settings that are used are:



- Post Processor Selected
- Knowledge Drawing
- DOIT file
- Current Material
- All Job related settings
- All Part related settings

Automatic G-Code Scan

The Automatic G-Code Scanner will check each NC file generated by Router-CIM Automation Suite for a specific post processor and if the code contains a value outside the limits set in the scanner, then an error will be displayed when the job is run, and the code will be placed into a separate folder so that it is not run by mistake and the error can be fixed.

Note: This feature is only available for machines that take industry standard G-Code and if 'Z0 is top of Material' is unchecked.



Automatic G-Code Scanner

 Add Machine  Delete Machine

Machine Name	Post	On/Off	X-Axis Limits Checked	X-Lower Limit	X-Upper Limit	Y-Axis Limits Checked	YLower	YUpper	Z-Axis Limits Checked	Z-Lower Limit	Z-Upper Limit
CIMTECH	RS274D.\$PP	<input checked="" type="checkbox"/>	No Checking	0	0	No Checking	0	0	Lower	-0.1	0

☐ Automatically Print Limit Check Report

Cancel Pending Edits

Save  Close 

You can give each machine set-up an name, and then select the post processor to be used for that set-up and change the X, Y, and Z settings and whether or not to scan each axis for problems.

 Add Machine  Delete Machine

Machine Name	Post	On/Off	X-Axis Limits Checked
CIMTECH	RS274D.\$PP	<input checked="" type="checkbox"/>	No Checking

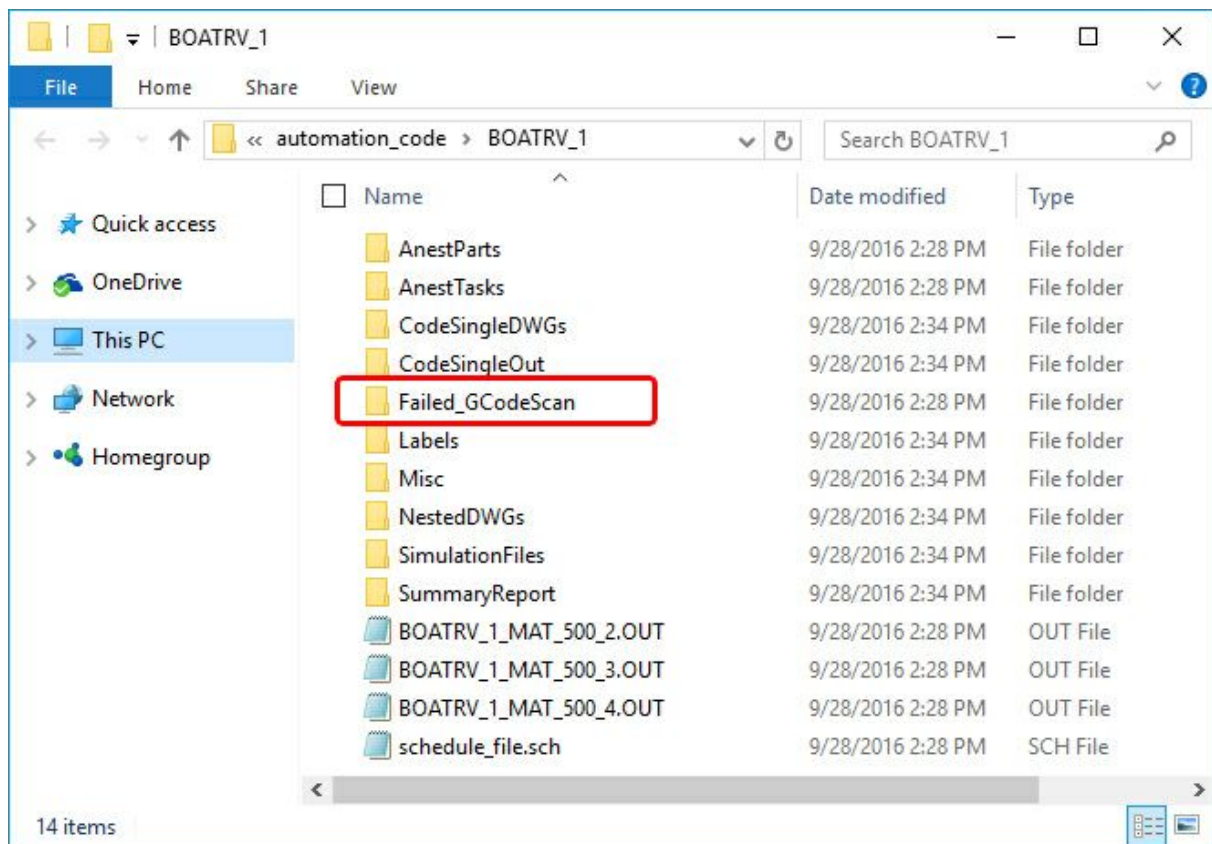
If after a job is run, there are values beyond the limits set you will see a warning in the Limit Check Results.

Limits Check Results					
Code File	Line #	Axis	Effective Value	Lower Limit	Upper Limit
BOATRV_1_MAT_250_1.OUT	22	Z	-0.25	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	24	Z	-0.15	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	25	Z	-0.26	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	29	Z	-0.25	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	31	Z	-0.15	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	32	Z	-0.26	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	36	Z	-0.25	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	38	Z	-0.15	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	39	Z	-0.26	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	43	Z	-0.25	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	45	Z	-0.15	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	46	Z	-0.26	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	50	Z	-0.25	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	52	Z	-0.15	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	53	Z	-0.26	-0.1	Not Set
BOATRV_1_MAT_250_1.OUT	57	Z	-0.25	-0.1	Not Set

In this case there were Z values more negative than the -0.1 value placed in the lower limit. All the files were flagged and the line number containing the error is shown along with its value.

At this point, you have the option to print this report by selecting the 'Print' button.

Once this window is acknowledged by selecting 'OK', the Results Folder can be opened and you will see a folder labeled Failed_GCodeScan and that will contain the NC code files that have failed the Automatic G-Code Scanner.



Migration Wizard

The Migration Wizard will transfer files from a Router-CIM Automation Suite backup that was created after installing a newer version into the same folder structure as the previous version.

MultiPost Default

MultiPost allows a user to have predefined job formats allowing a single job to be processed multiple times in order to get all the results needed when programming for multiple CNC machines including machine dependent NC code.

For more information on setting up MultiPost, go to the [MultiPost](#) section.

NCVAR File Editor

The NCVAR File Editor allows you to change the NCVARs (NC Variables) that control certain items in the Router-CIM software.

NOTE: Adding/Modifying any variable affects how Router-CIM and Router-CIM Automation Suite will perform and process the parts. Adding/Modifying any variable, including the ones mentioned here, should be done with extreme care. To add/modify

any variable not mentioned here, please consult CIM-Tech prior to any addition/modification.

Changes include:

[Change a Variable](#)

[Add a New Variable](#)

[Delete a Variable](#)

[Add a Note to a Variable](#)

[Delete a Note to a Variable](#)

For more information on the types of NC variables that can be changed or added, click [here](#).

NCVar file Editor

NCVar Section:

- Geometric (Metric)
- Geometric (Inch)
- Layer
- System
- Other

NCVar List

NCVar Edit

Name:

Value:

Notes

New Var Delete

New Row Delete Row

Save Settings Exit

How to Change a Variable using the NCVAR File Editor:

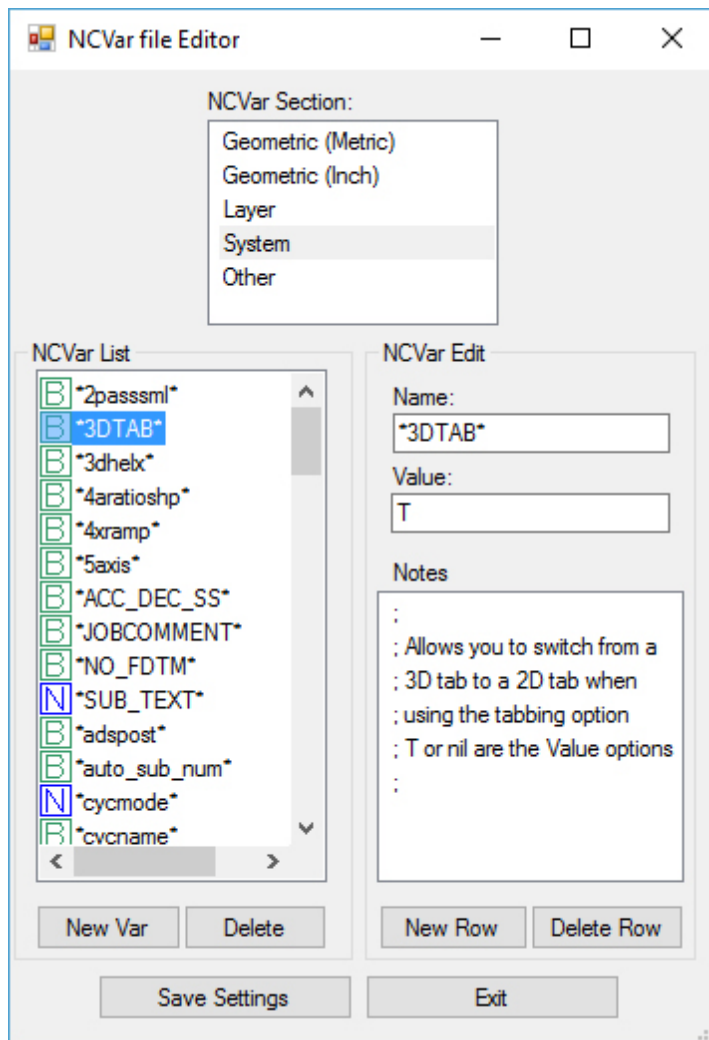
To change a variable using the NCVAR File Editor, you will first need to find the variable you would like to change under the appropriate section.

For more information on the types of NC variables that can be changed or added, click [here](#).

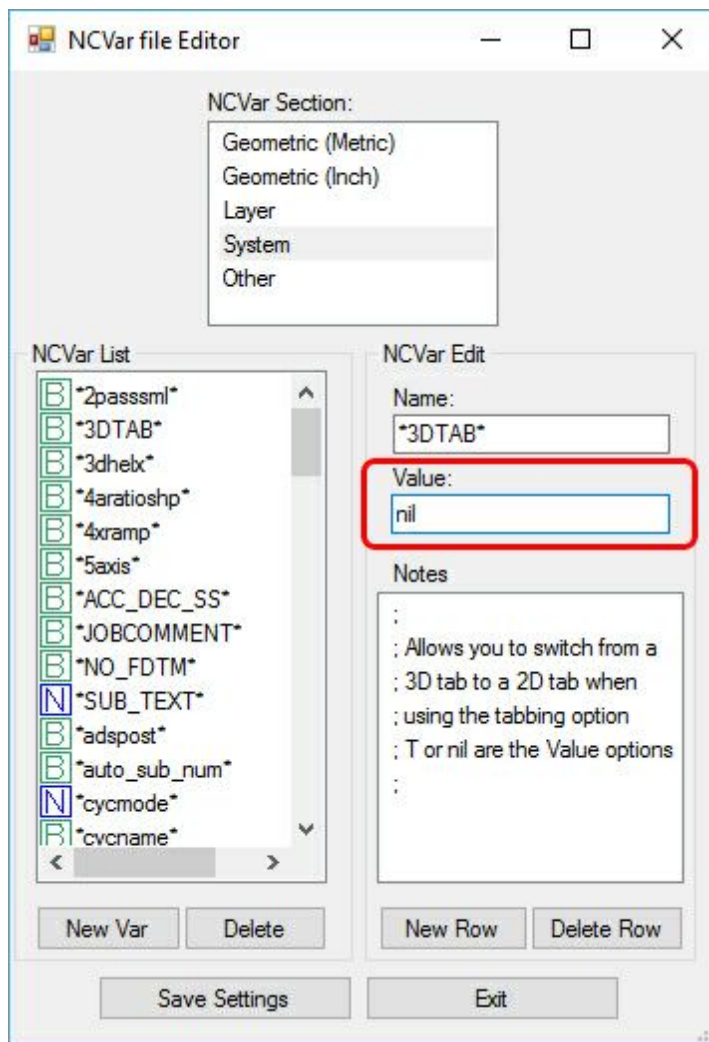
The sections available are:

- Geometric (Metric)
- Geometric (Inch)
- Layer
- System
- Other

Once the variable is located, select on the variable name in the left hand column. For this example, we will use the *3DTAB* variable under the System section:



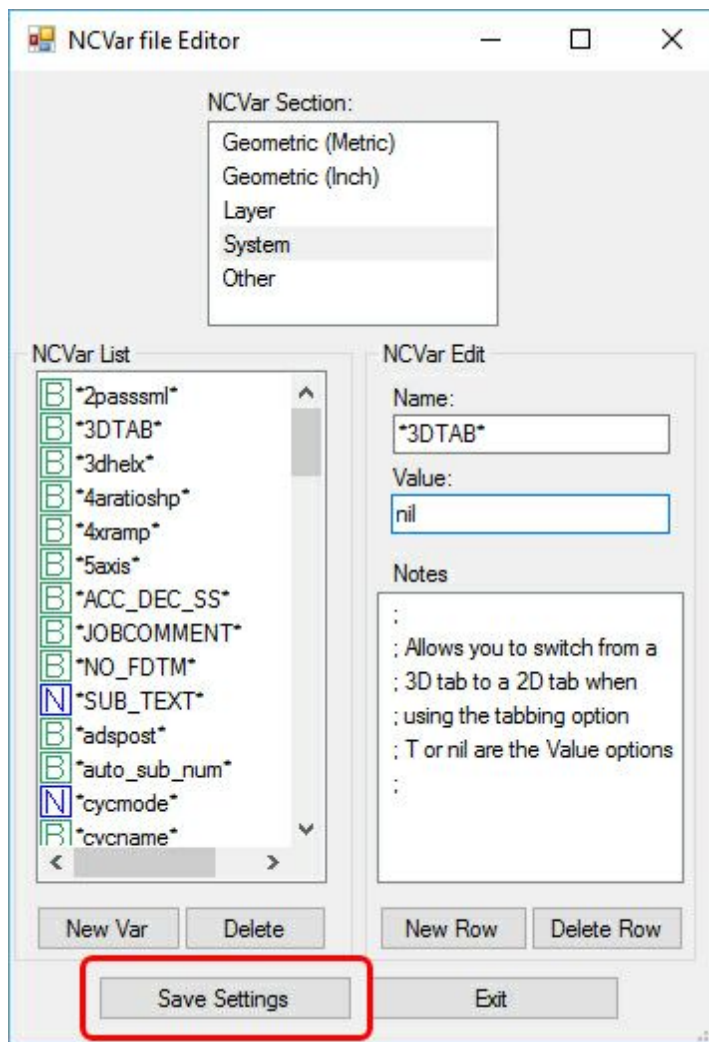
To change the value, select the Value field in the right column and type in the new value you would like to use:



Once you have entered the appropriate Value in this field, make sure to select on a different field such as Name in order for the file to be updated.

When all the changes have been made that are needed, simply select the 'Save Settings' button and the changes will be saved:

Note: Selecting the 'Save Settings' button will also close the NCVAR File Editor



If you do not want to save your changes, simply select the 'Exit' button and the NCVAR File Editor will close without saving any changes.

How to Add a Variable using the NCVAR File Editor:

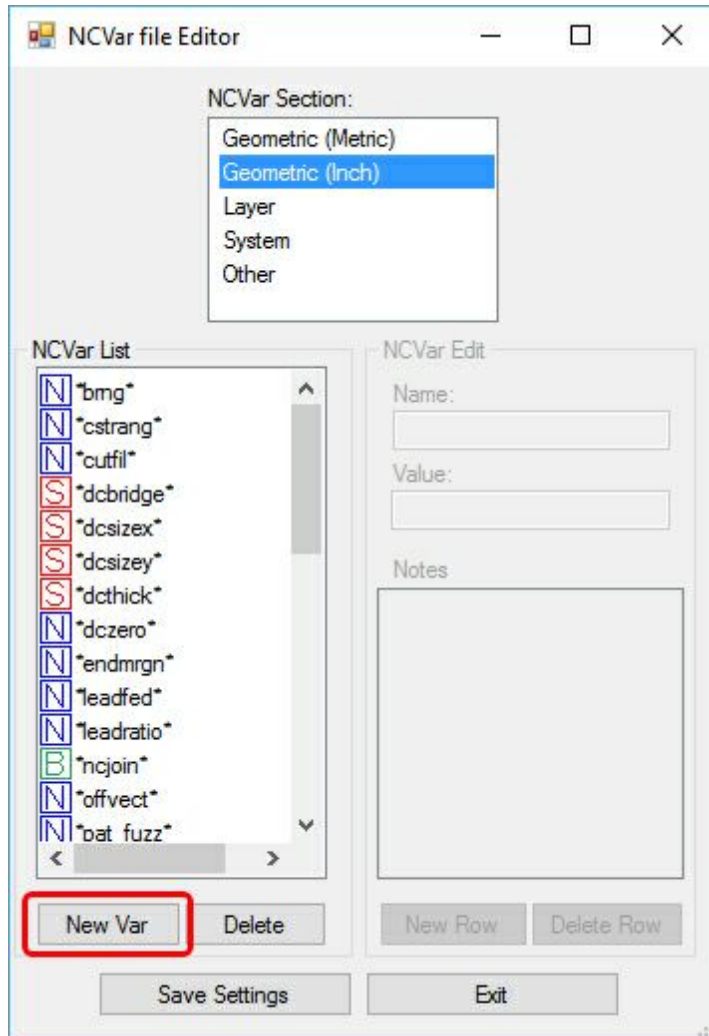
To add a variable using the NCVAR File Editor, you will first need to select the appropriate section to add the variable to.

For more information on the types of NC variables that can be changed or added, click [here](#).

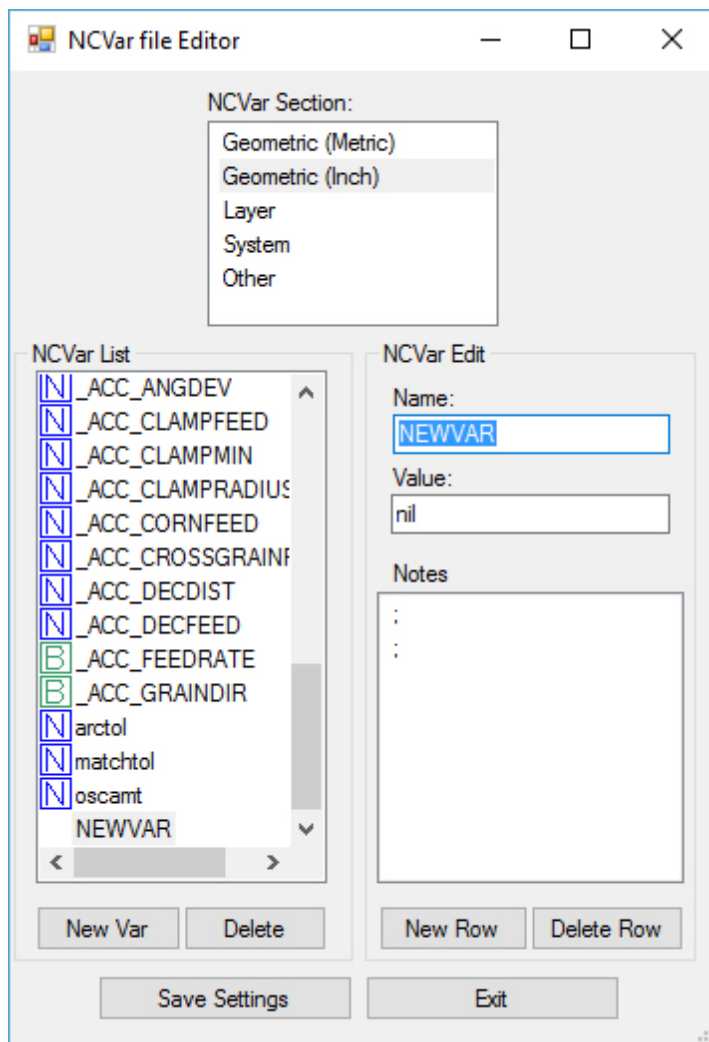
The sections available are:

- Geometric (Metric)
- Geometric (Inch)
- Layer
- System
- Other

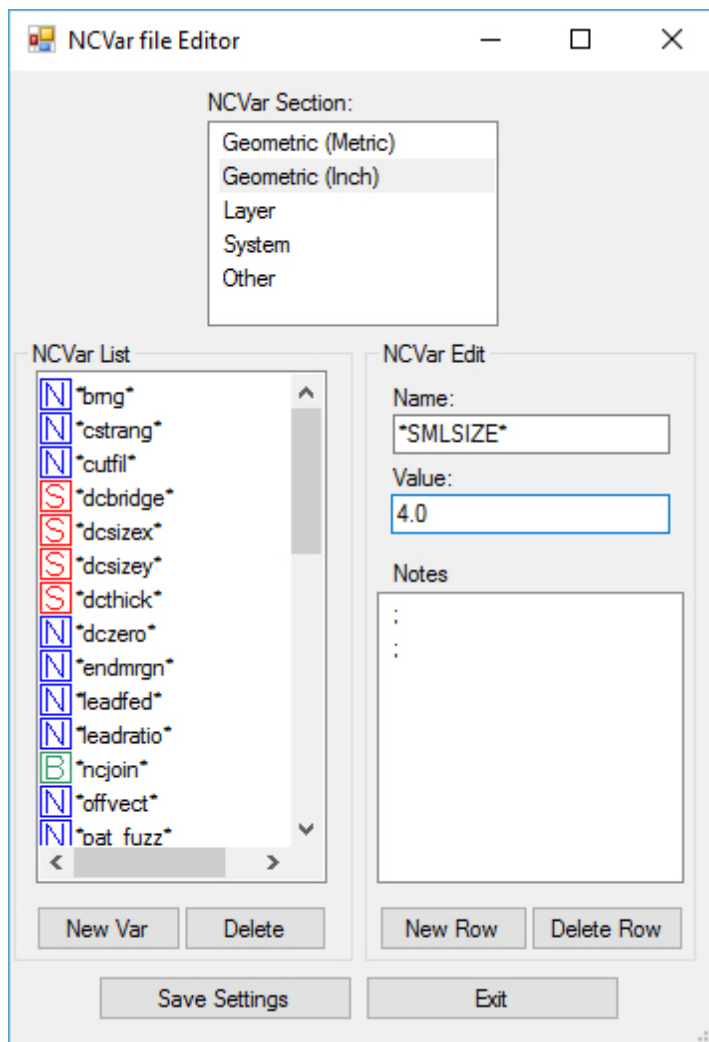
Once the section is selected, select on the 'New Var' button. For this example, we will use the *SMLSIZ* variable under the Geometric (Inch) section:



When the 'New Var' button is selected, you will see the right column become active:



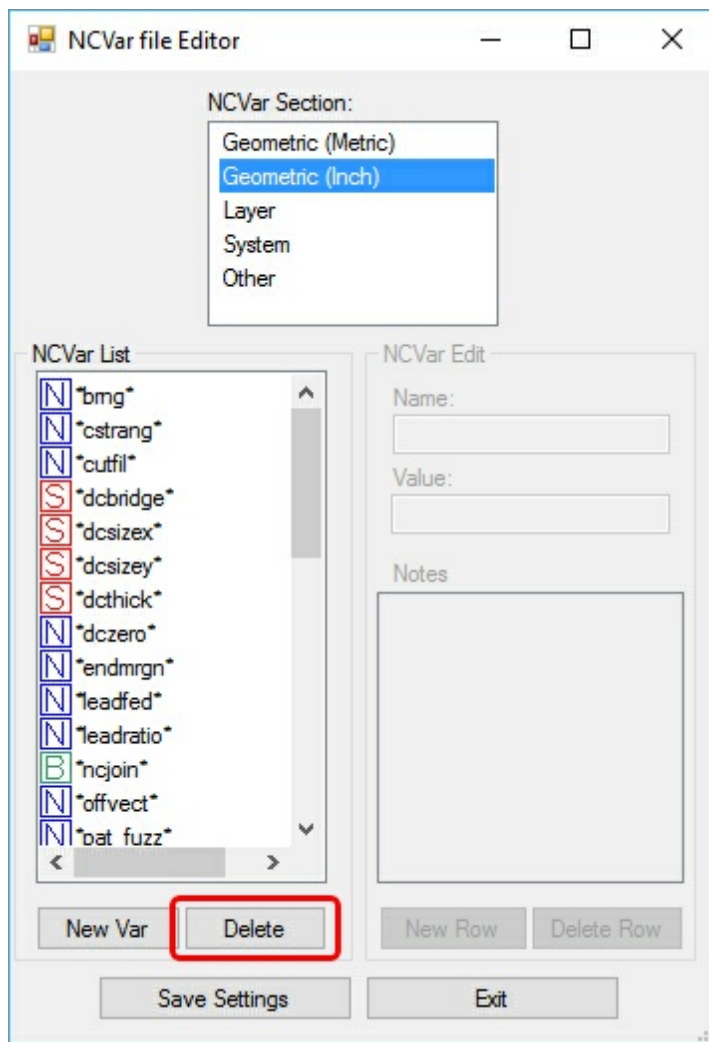
Type in the name of the variable you want to add exactly like it is stated in help manual and add the value that you want the variable to be set to:



Once you have completed entering the variable name and value, select the 'Save Settings' button in order for it to be added to the list.

How to Delete a Variable using the NCVAR File Editor:

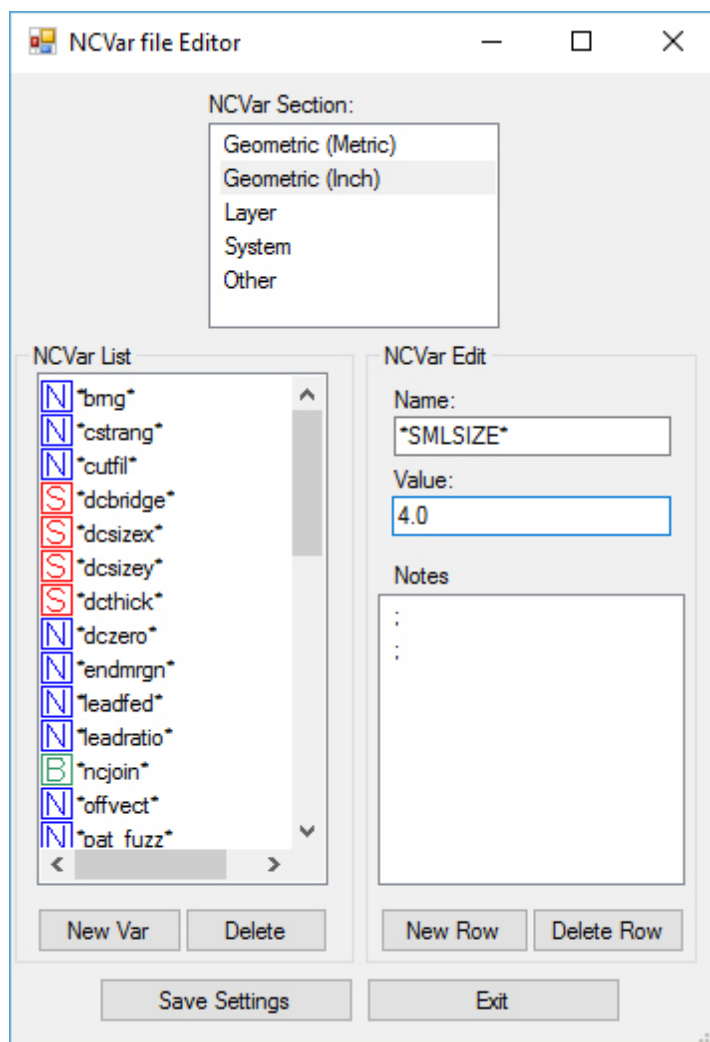
To delete a variable using the NCVAR File Editor, you will first need to select the appropriate section that has the variable you want to delete. Highlight the variable and select the 'Delete' button:



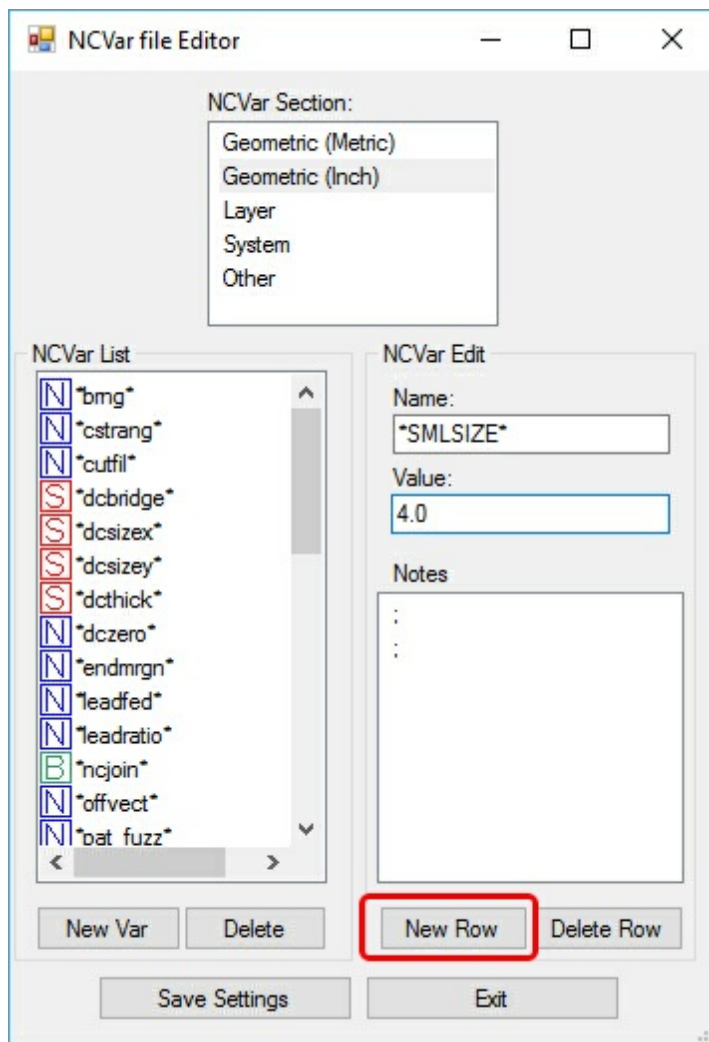
How to Add a Note to a Variable using the NCVAR File Editor:

When adding an NCVAR variable, take care in including a Note so that it can be referred to later for more information.

To add a note to a variable, you must first select the variable you want to add the note to:

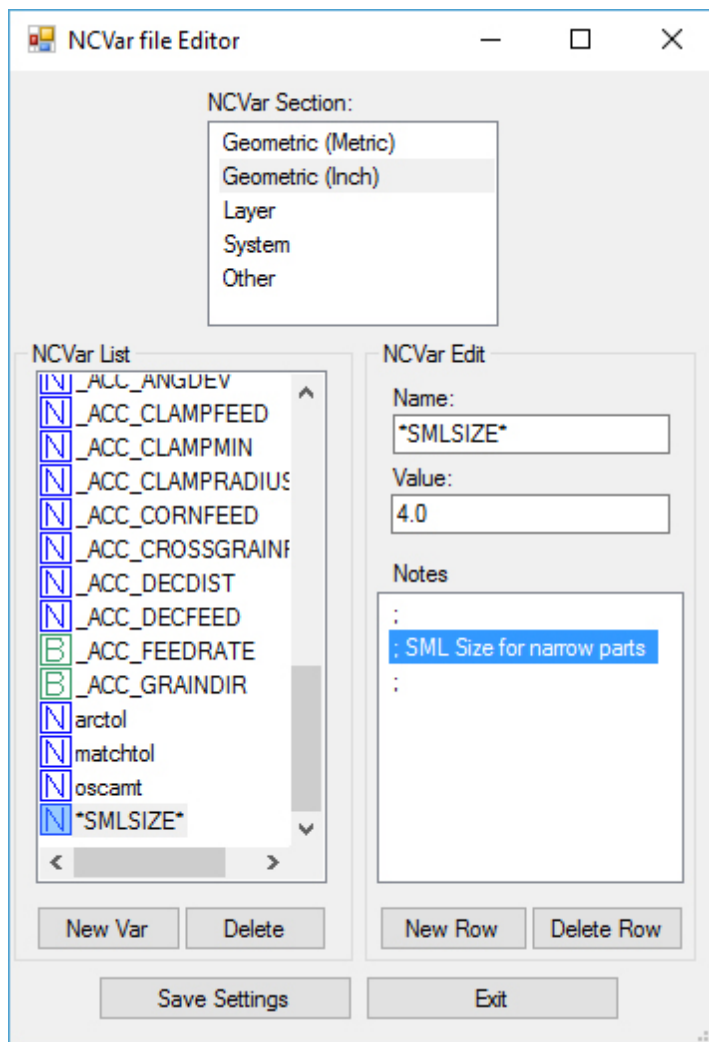


Highlight the semicolon in the row that you want to add a note **UNDER** and select the 'New Row' button:



This will add a row under where you have selected. Select on this row to edit by doing a slow double click:

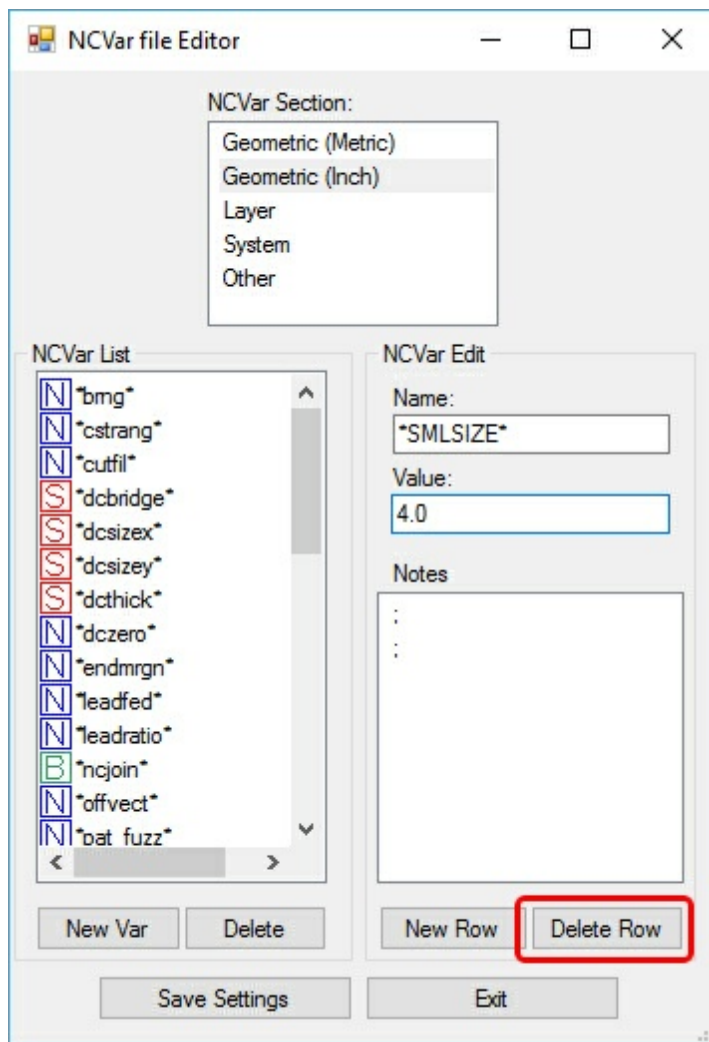
Note: When adding a note to a row, **DO NOT** delete the semicolon.



When the appropriate note has been added, select the 'Save Settings' button and your note will be entered.

How to Delete a Note to a Variable using the NCVAR File Editor:

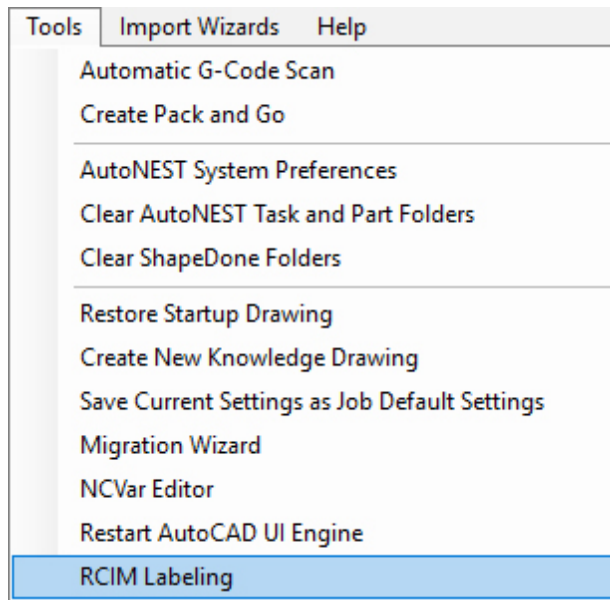
If you need to delete a row from the Notes section, highlight the row and select the 'Delete Row' button:



When the appropriate note has been deleted, select the 'Save Settings' button and your note will be changed.

Label Designer

This Label Designer allows you to make a template of where you want part properties to appear on a printed label.



The **Label Designer** lets you create custom labels that can be individually created or designed to match a sheet template that you have created or to create individual labels for use with the Touch-N-Print labeling system or a stand-alone printing system.

Note: If creating labels for a sheet template, the size of the label must match exactly to the label size determined by the Sheet Layout.

There are 5 different labels that you have access to:

- 1) [Part Label](#)
- 2) [Job Label](#)
- 3) [Material Label](#)
- 4) [Scrap Label](#)
- 5) [NC Code Label](#)

To create a custom label using the Label Designer, click [here](#).

Select a label listed above for more information.

Once your labels have been designed and/or your sheet layout templates, you will create a configuration or multiple configurations that you want for your labels when a job is run in Router-CIM Automation Suite.

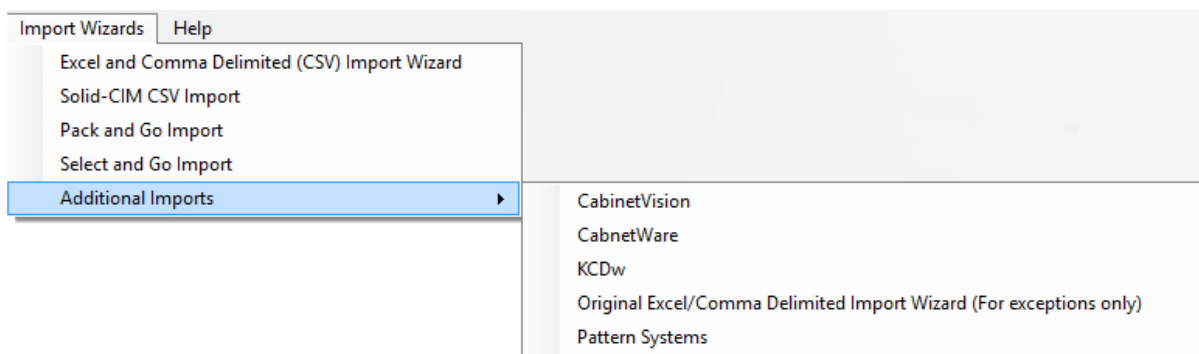
For information setting up a configuration for use in the job, click [here](#).

Single Part Report Configuration

Adjusting the values in the Single Part Report Configuration window will allow you to more accurately create Single Part Reports if this option is selected under the '[Printing and Labels](#)' tab.

Active	Machine Name	Post	EstQtyPerRun	EstYearlyQty	PCSSkid	HandlingSkid	PCSOperation	SetupTimePerRun	MachineRate	Material
<input checked="" type="checkbox"/>	DefaultMachine		100	1000	50	2	1	10	125	0

Import Wizards Menu



Router-CIM Automation Suite has several Import Wizards used to create jobs in just a few steps. There are several 3rd party packages that create geometry and job settings that Router-CIM Automation Suite can read and then create a job from those parts and/or settings in the 'Additional Imports' fly-out menu.

In addition to the 3rd party imports, there are two generic imports for either a comma-delimited file or an Excel spreadsheet that can be handy to create jobs from. Additionally, there is the Pack and Go Import to bring a job into Router-CIM Automation Suite that was packaged previously using the Pack and Go option.

The Import Wizards available are:

- [Excel and Comma Delimited \(CSV\) Import Wizard](#)
- [Solid-CIM CSV Import](#)
- [Pack and Go](#)
- [Select and Go Import](#)

Additional Imports

- [CabinetVision](#)
- [Cabnetware](#)
- [KCDw](#)
- [Original Excel/Comma Delimited Import Wizard \(For exceptions only\)](#)

- o [Pattern Systems](#)

Note: Additional Imports are based on specific versions of the third party software.

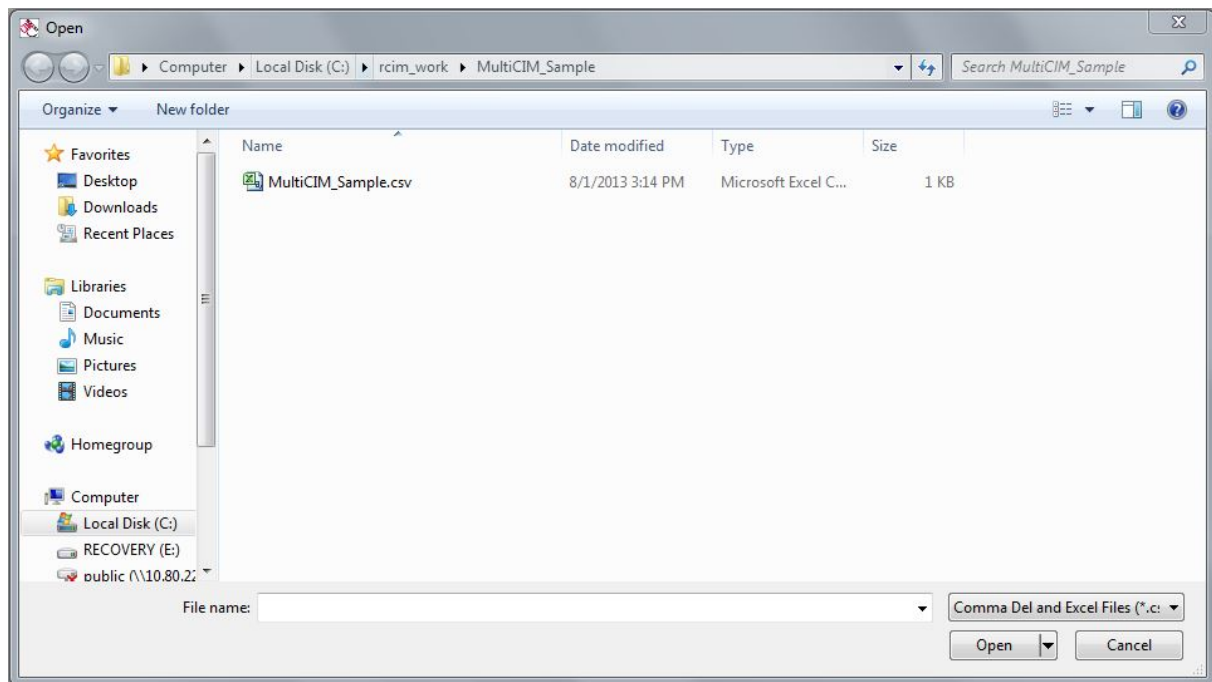
Excel and Comma Delimited (CSV)

The Excel and Comma Delimited (CSV) Import Wizard allows you to import parts from a file with an excel or comma-delimited format.

Note: In order to use an Excel or Comma Delimited (CSV) to import Assemblies/Jobs instead of parts, please click [here](#) for more information.

Select File to Import

When you select the Excel and Comma Delimited (CSV) Import Wizard, a dialog box will appear where you can select an Excel or .CSV file to import into Router-CIM Automation Suite.



Once you have selected a file, Router-CIM Automation Suite will open the Import Wizard allowing you to format the import wizard based on your file setup.

Import Wizard 2.0

Optional Features

Select a saved format Part Material Field Selection

☐ Material Code ☒ Material Description

Number of header rows

Column 1	Column 2	Column 3	Column 4	Column 5	Column 6	Column 7
Select Data Type	Select Data Type	Select Data Type	Select Data Type	Select Data Type	Select Data Type	Select Data Type
Column1	Column2	Column3	Column4	Column5	Column6	Column7
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\RIGHT_UPPER.dwg	MAHOGANY_0...	1	22.5000 x 14.25...		22.5000	14.2500
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\RIGHT_BOTTOM.dwg	MAHOGANY_0...	1	22.5000 x 14.25...		22.5000	14.2500
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\LEFT_BOTTOM.dwg	MAHOGANY_0...	1	22.5000 x 14.25...		22.5000	14.2500
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\DRAWER_RIGHT_BASE.dwg	MAHOGANY_0...	1	4.0000 x 14.250...		4.0000	14.2500
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\RIGHT_SIDE.dwg	MAHOGANY_0...	1	29.0000 x 22.50...		29.0000	22.5000
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\RIGHT_INNER.dwg	MAHOGANY_0...	1	29.0000 x 22.50...		29.0000	22.5000
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\LEFT_SIDE.dwg	MAHOGANY_0...	1	29.0000 x 22.50...		29.0000	22.5000
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\DRAWER_RIGHT_BOTTOM.dwg	MAHOGANY_0...	1	7.7500 x 14.125...		7.7500	14.1250
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\DRAWER_RIGHT_MIDDLE.dwg	MAHOGANY_0...	1	7.7500 x 14.125...		7.7500	14.1250
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\DRAWER_RIGHT_TOP.dwg	MAHOGANY_0...	1	7.7500 x 14.125...		7.7500	14.1250
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\BACK.dwg	MAHOGANY_0...	1	70.7500 x 29.00...		70.7500	29.0000
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\DESK_TOP.dwg	MAHOGANY_1...	1	72.0000 x 24.00...		72.0000	24.0000
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\DRAWER_LEFT_BASE.dwg	MAHOGANY_0...	1	4.0000 x 14.250...		4.0000	14.2500
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\DRAWER_LEFT_BOTTOM.dwg	MAHOGANY_0...	1	7.7500 x 14.125...		7.7500	14.1250
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\DRAWER_LEFT_MIDDLE.dwg	MAHOGANY_0...	1	7.7500 x 14.125...		7.7500	14.1250
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\DRAWER_LEFT_TOP.dwg	MAHOGANY_0...	1	7.7500 x 14.125...		7.7500	14.1250
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\LEFT_INNER.dwg	MAHOGANY_0...	1	29.0000 x 22.50...		29.0000	22.5000
C:\rcim_work\SCIMGRAIN_MATCH_ASSEMBLY\LEFT_UPPER.dwg	MAHOGANY_0...	1	22.5000 x 14.25...		22.5000	14.2500

Assign Column Headings

Assigning Column Headings will allow you to select the data for each column in the imported file. The choices available in the pulldown list are:

Note: Part Name, Part Material and Quantity must be assigned to a column in order for Router-CIM Automation Suite to import a file.

Variable	Field Option	Description	Value Accepted
Ignore	Optional	Ignores entire column during the import process	N/A
Part Name	Required	The 'Part Name' variable will define the location of the part on the computer. This field should be reserved for the file path location.	File Path Location
Part Material	Required	This field is used for defining the material of the part. For a seamless transition through the RCIM import wizard, the material used should match the material description in the RCIM Materials Database.	Material Description
Quantity (Qty)	Required	This field is used for defining the number of parts needed.	Whole Number
X Dim	Macros Only	Defines the XDIM variable with use in a parametric macro	Real Number, 30 Character Maximum

Y Dim	Macros Only	Defines the YDIM variable with use in a parametric macro	Real Number, 30 Character Maximum
Z Dim	Macros Only	Defines the ZDIM variable with use in a parametric macro	Real Number, 30 Character Maximum
Ignore Panel	Macros Only	This variable controls if the 'Panel' layer of a parametric macro is ignored during the automation process	0 (No) or 1 (Yes)
Backside	Macros Only	This variable controls if an individual parametric macro has an associated backside macro	0 (No) or 1 (Yes)
Description	Optional	This variable controls the information included under the 'Record Desc' field located in the part properties window. Each part can be imported with a unique description for labeling purposes	80 Character Maximum
Label Info 1	Optional	This variable controls the information included under the 'Label Field 1' located in the part properties window. Each part can be imported with unique label information.	25 Character Maximum
Label Info 2	Optional	This variable controls the information included under the 'Label Field 2' located in the part properties window. Each part can be imported with unique label information.	25 Character Maximum
Label Info 3	Optional	This variable controls the information included under the 'Label Field 3' located in the part properties window. Each part can be imported with unique label information.	25 Character Maximum
Label Info 4	Optional	This variable controls the information included under the 'Label Field 4' located in the part properties window. Each part can be imported with unique label information.	25 Character Maximum
Label Info 5	Optional	This variable controls the information included under the 'Label Field 5' located in the part properties window. Each part can be imported with unique label information.	99 Character Maximum
Label Info 6	Optional	This variable controls the information included under the 'Label Field 6' located in the part properties window. Each part can be imported with unique label information.	99 Character Maximum
Label Info 7	Optional	This variable controls the information included under the 'Label Field 7' located in the part properties window. Each part can be imported with unique label information.	99 Character Maximum

Label Info 8	Optional	This variable controls the information included under the 'Label Field 8' located in the part properties window. Each part can be imported with unique label information.	99 Character Maximum
Rotate Part	Optional	This variable controls the part option to rotate the individual part. The rotation of the part occurs once the job has started to be processed and prior to the processing of the actual part itself.	0 (No) or 1 (Yes)
Rotate Angle	Optional	This variable controls the amount of rotation (1 degree increments) when the Rotate Part option is selected	Whole Numbers, 0 – 360
Knowledge Drawing	Optional	This variable defines the 'Knowledge Drawing' that will be used to process the part. This option can be different per part when 'Code As Single Part Only' is selected in the part properties window. This variable must be the same as the other parts within a job when nesting.	30 Character Maximum (must include .dwg)
DOIT File	Optional	This variable defines the 'DOIT File' that will be used to process the part. This option can be different per part when 'Code As Single Part Only' is selected in the part properties window. This variable must be the same as the other parts within a job when nesting.	30 Character Maximum (Must include .dat)
Post Processor	Optional	This variable defines the 'Post Processor' that will be used to process the part. This option can be different per part when 'Code As Single Part Only' is selected in the part properties window. This variable must be the same as the other parts within a job when nesting.	30 Character Maximum (must include .\$pp)
Print Nests	Optional	This variable controls the option for printing the created nested sheet after a job has processed.	0 (No) or 1 (Yes)
Print Single Parts	Optional	This variable controls the option for printing the single part after a job has processed when 'Code as Single Part Only' is selected.	0 (No) or 1 (Yes)
Job Name	Optional	This variable controls the name of the job created in Router-CIM Automation Suite	30 Character Maximum. NO SPACES.
Filler Quantity (Filler Qty)	Optional	This variable controls the amount of filler parts that are available to nesting per the particular part	4 Numeric Character Maximum or -1
Start Point Longest Side	Optional	This variable controls the option to have the start point of the cut cycle be situated on the longest side of a particular piece of geometry for the particular part	0 (No) or 1 (Yes)

Nest Rotation	Optional	This variable controls the rotation of the part within the material that it will be nested with. The part and the material can have defined nest rotation variables that allow for nesting to handle the part different than what is defined on the nesting material	Same As Material; ALL; 0; 0 90; 0 90 180
Mirror	Optional	This variable controls whether a part will be the mirror image. Direction of the mirror is controlled by the variable 'Mirror Type'.	0 (No) or 1 (Yes)
Mirror Type	Optional	Defines the axis on which the part will be mirrored	Horizontal or Vertical
Veneer Match Name	Optional	Name defined by the user to identify parts that need to be nested together	30 Character Maximum. NO SPACES.
Veneer Match Location Point	Optional	Location Point needed for the Veneer Matching feature in Router-CIM Automation Suite	X,Y Coordinate point. NO SPACES
Veneer Match Rotation	Optional	Veneer Match Rotation Field	Numeric value up to 3 numbers
Packing Direction	Optional	Defines the direction that the nesting engine will fill up the sheet with the parts defined by the job.	3 (Auto) 4 (Horizontal) 5 (Vertical)
Nesting Start	Optional	Defines the corner that the nesting engine will start to fill up the sheet with the parts defined by the job.	1 (Lower Left) 2 (Upper Left) 3 (Upper Right) 4 (Lower Right)
Sheet Origin	Optional	Defines the corner of the sheet that the output code files will base the X and Y axis direction on. Ex. 1 would tell the code to be based off the lower left of the sheet meaning positive X and positive Y.	1 (Lower Left) 2 (Upper Left) 3 (Upper Right) 4 (Lower Right)
NC Code Extension	Optional	This will be the file extension that the output files will be created with. The extension is the only thing needed. You do not have to define the decimal point. Ex. TXT, CNC, OUT, etc.	Any 2 or 3 digit file extension that your CNC accepts
Output to Saw	Optional	This variable controls the 'Output to Saw' check box is checked.	0 (No) or 1 (Yes)
Variable	Optional	User defined fields for additional parametric macro variables	

Save Format (header rows and column headings)

You are able to save the data that is set up for a job by selecting '**Save Format (header rows and column headings)**'. You will be prompted for a name of the Saved Format:

Selecting OK will add this name to the Saved Format list so that you can select this format at a later time instead of re-selecting each column.

Delete Selected Format

You may delete the saved selected format from the saved format list.

Part Material Field Selection

This option allows you to select whether you want Router-CIM Automation Suite to match the material in your excel or comma delimited file based on the material description or material code that you have set up in the Router-CIM Automation Suite Materials database.

Clear Current Column Format

If you select a saved format but need to start from the default column headings, select the '**Clear Current Column Format**' button to set the column headings to the default.

Number of Header Rows

If your comma delimited file has header rows in it for any reason, you can bypass those and get straight to the data by specifying how many header rows to skip.

Import

Finally when you are done selecting your file and headings, you can import the file into Router-CIM Automation Suite and it will build a job from the selected data

Importing an Excel and Comma Delimited (CSV) to assign TAGGED variables within a Parametric Macro

If you are using an Excel or Comma Delimited file to change TAGGED variables in a parametric macro, you will need to format a column in the excel file to identify the variable name and equal it to the value you are needing.

J
STILE_RIGHT=3.0

When creating an import wizard format, you will set this column to 'Variable' from the drop-down options.

Importing an Excel and Comma Delimited (CSV) to Create an Assembly Job

If you are using an Excel or Comma Delimited file to reference Jobs in Router-CIM Automation Suite instead of a defined part, you will need to create a Macro Format Job import file (mjf).

To create this, it is simply identifying the job in the part name column of your Excel or Comma Delimited file followed by .MJF

	K	L
1	Part Name	Quantity
2	CIMTech-DT-STD-Package.mjf	1

Once the Excel or Comma Delimited file has been defined, please refer to the beginning of this section and create an 'Import Wizard Format' to identify the columns needed for import.

When using an Excel or Comma Delimited file to import an assembly job already created in Router-CIM Automation Suite, you will need to define the 'Assembly Folder'. This will be a folder created in the Job Library that will contain the assembly jobs you create and will be calling from the MJF import file.

Name	Created On
Job Library	
MACRO_ASSEMBLIES	11/6/2020 12:00:00 AM
CIMTech-DT-STD-Package	11/4/2020 1:14:11 PM

For more information on setting the 'Assembly Folder', please refer to the ['Import Wizard Settings'](#) section.

When using an Excel or Comma Delimited file to create an assembly job, once imported, the job will look like this:

Part #	PartName	Material	Qty
1	CIMTech-DT-STD-Package	CIMTech-DT-STD-Package	1

Solid-CIM CSV

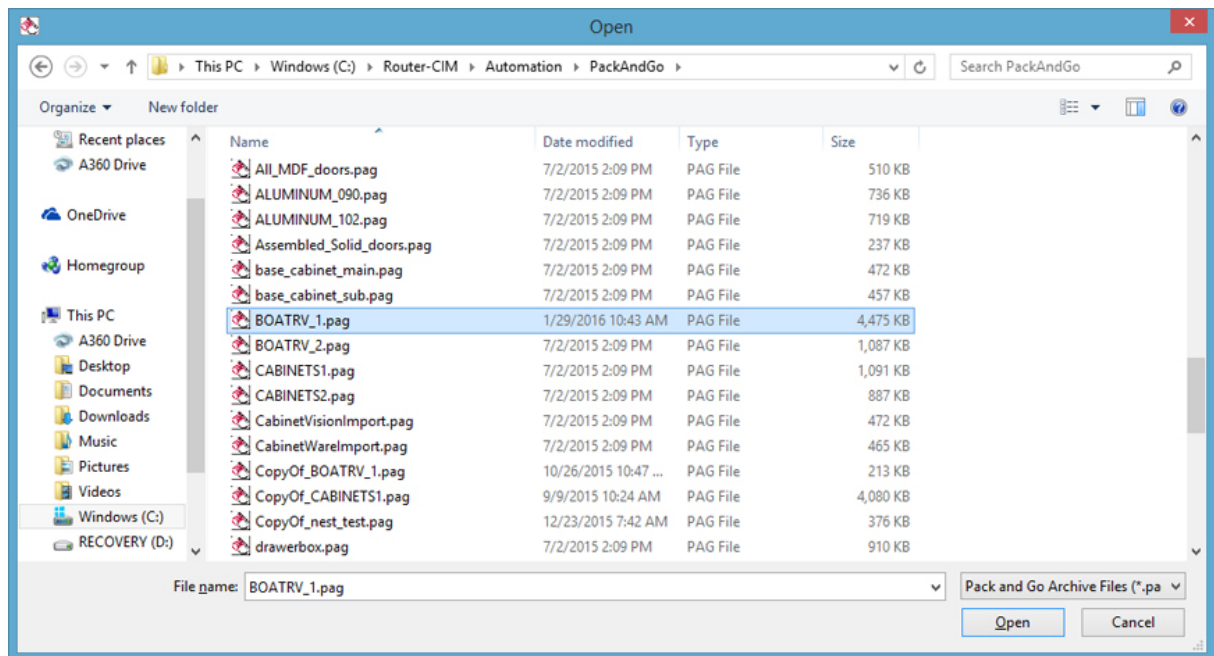
The Solid-CIM CSV Import Wizard allows you to import parts from the CSV file that was created by the Solid-CIM 3D add-on.

Column	Name	Description	Field Associated
A	Drawing Name	Fully qualified part drawing name.	Part Name
B	Material	Material Name	Part Material
C	Quantity	The number of times this part appears in the assembly.	Part Quantity
D	Description	Part description in the form of Length X Width X Thickness	Label Info 1
E	Stock	Router-CIM material code	Ignore
F	Length	Length of part in X	X Dim
G	Width	Width of part in Y	Y Dim
H	Thickness	Height of part in Z	Z Dim
I	Is Rectangular	1 = part is rectangular 0 = non-rectangular	Ignore
J	Has Top Features	1 = part has top features 0 = no features on top	Ignore
K	Has Side Features	1 = part has side features 0 = no features on sides	Ignore
L	Has Bottom Features	1 = part has bottom features 0 = no features on bottom	Ignore
M	Part Name	Unqualified part drawing name.	Description

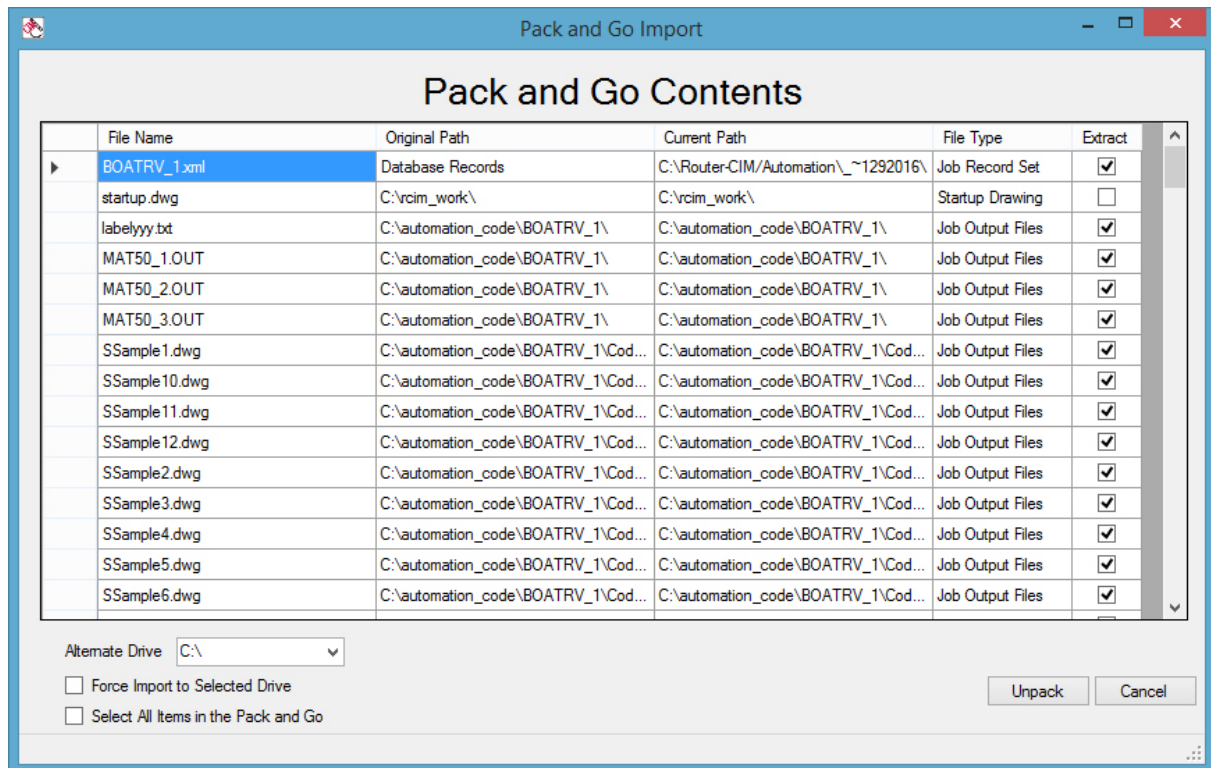
N	Part Number	Part number (Row number in File)	Ignore
O	Number of Variables	Number of variable columns to follow	Select Data Type
P...	User Variables	Each column is a user variable value. Only developed by special programs.	Field 16
P + Vars + 1	Appearance Recognition	0 = False, 1 = True	Field 17
Next	Number of EdgeBand Definitions	Number of Unique EdgeBand type definitions to follow	Field 18
Next	EdgeBand Type	EdgeBand Type	Field 19
Next	EdgeBand Description	EdgeBand Description	Select Data Type
Next	EdgeBand Offset	EdgeBand Offset	Select Data Type
Next	EdgeBand Total Length	EdgeBand Total Length	Select Data Type
Repeated	4 Columns for each Unique EdgeBand	EdgeBand Definition	Select Data Type

Pack and Go

Pack and Go is the Router-CIM Automation Suite format for packaging all the elements of a job into one file. To import that file into Router-CIM Automation Suite as a job, select pack and go from the import list, and a window will appear where you can select the PAG file from.



Once that file has been selected, you will be show a list of all the elements of a job that can be imported. Most of the time, you can select all the elements, but a check box is provided for you to remove elements from the selection.



Once you have all the elements selected that you wish to import, select the 'Unpack' button and a job will be created in the current folder.

Select and Go

Select and Go is an import wizard that allows you to select a CSV or Excel file and import that file as a job in Router-CIM Automation Suite.

The difference from using the Excel and Comma Delimited Import Wizard is that Select and Go will skip the import process and use the defaults that you have set up in File/Settings under the [Import Wizard Settings tab](#) to create a job in Router-CIM Automation Suite.

You can then review the job prior to selecting the **'Run Job'** button.

Additional Imports

The following section talks about the additional Import Wizards that are available through Router-CIM Automation Suite.

Note: Additional Imports are based on specific versions of the third party software. Please check your version to see if you are able to use these import wizard options.

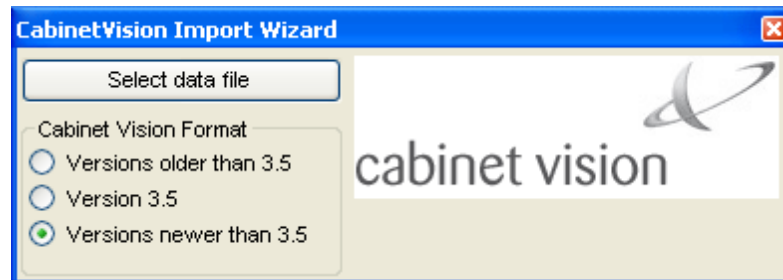
Additional Imports

- [CabinetVision](#)
- [Cabnetware](#)
- [KCDw](#)
- [Original Excel and Comma Delimited \(CSV\) Import Wizard](#)
- [Pattern Systems](#)

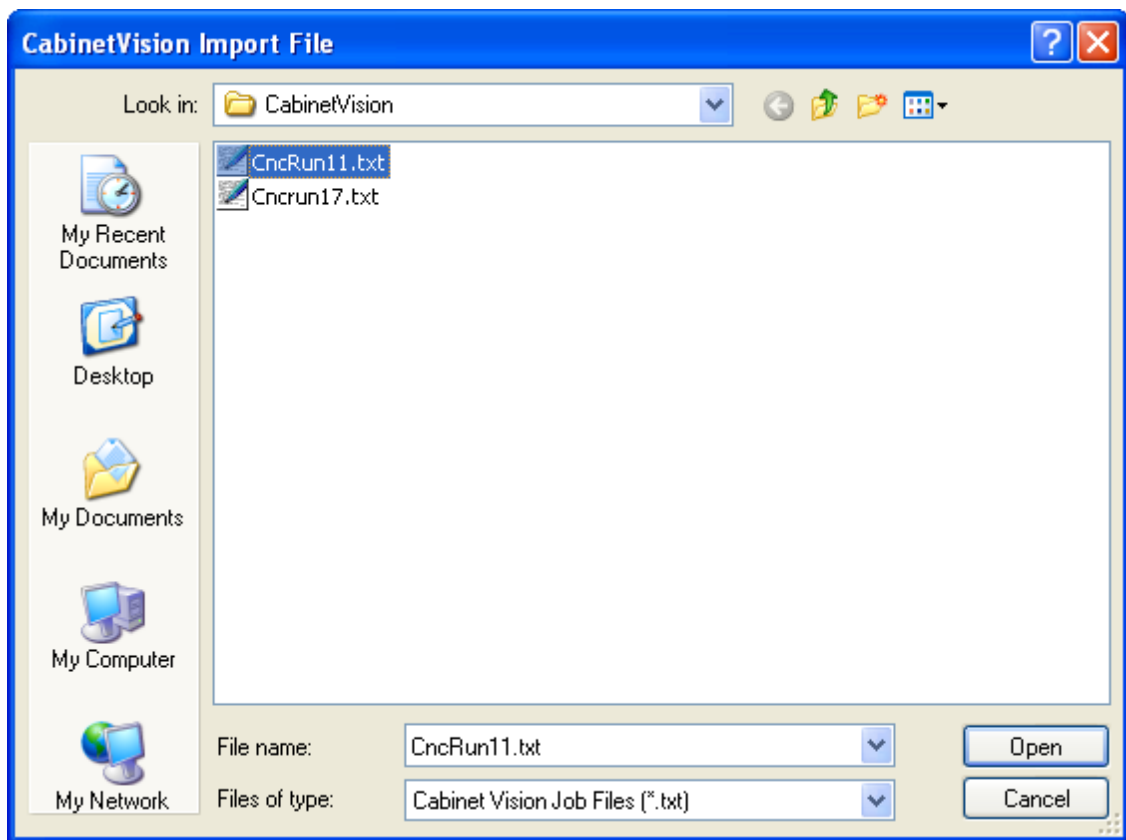
CabinetVision

Cabinet Vision is a Planit Solutions cabinet design and manufacturing program. This program has modules that allow it to export layered DXF files that can be imported into Router-CIM Automation Suite with the help of the Cabinet Vision import wizard.

After making a job in Cabinet Vision, select the Cabinet Vision Import Wizard. You can select the format that matches the version of your Cabinet Vision software.



A window will appear allowing you to select the data file that was produced along with the layered DXF files.



Select the Cncrun file for the job you wish to run and click Open. The files will be imported into Router-CIM Automation Suite in the current Jobs folder. The Cncrun file contains all the data for the job relating to the parts, materials, quantities and label information.

Settings for Cabinet Vision

The following are the default layer names that should be used in Cabinet Vision.

- The outside of a part should always be placed on a layer that begins with PANEL
- All vertical drill holes, regardless of size, should be placed on layer DRILL
- The Board layer should be placed on a layer that begins with BOARD
- Dadoes and rabbets with a width of .25 inch should be rectangles on layer ROUTE 250
- Dadoes and rabbets with a width of .375 inch should be rectangles on layer ROUTE 375
- Dadoes and rabbets with a width of .500 inch should be rectangles on layer ROUTE 500
- Dadoes and rabbets with a width of .625 inch should be rectangles on layer ROUTE 625
- Dadoes and rabbets with a width of .750 inch should be rectangles on layer ROUTE 750

Layer	Knowledge	Tool	Spindle#
Panel	Panel	.5" Compression	1
Drill	Drill	Multi Spindle Gang Drill	Varies
Route250 Route375	Dado 250	.25 Router Bit	2
Route500			

Route625 Route750	Dado 500	.5" Down Shear	3
----------------------	----------	----------------	---

Table 1. Layer to Knowledge Associations for Cabinet Vision Router-CIM

Cabinet Vision Notes:

All geometry that is to be cut in a like fashion (same tool, spindle, feedrate, etc.) should be placed on the same layer.

Typically, dadoes and rabbets should be output as rectangles that overlap the edge of the material by one tool radius (if the cut is to extend to the edge of the part).

All geometry should be output with negative thickness indicating cut depth.

Backside Parts with Cabinet Vision:

There is a provision in Cabinet Vision to identify the parts which have backside or secondary operations. Likewise in Router-CIM Automation Suite there is a similar provision to cut them apart from the nested parts.

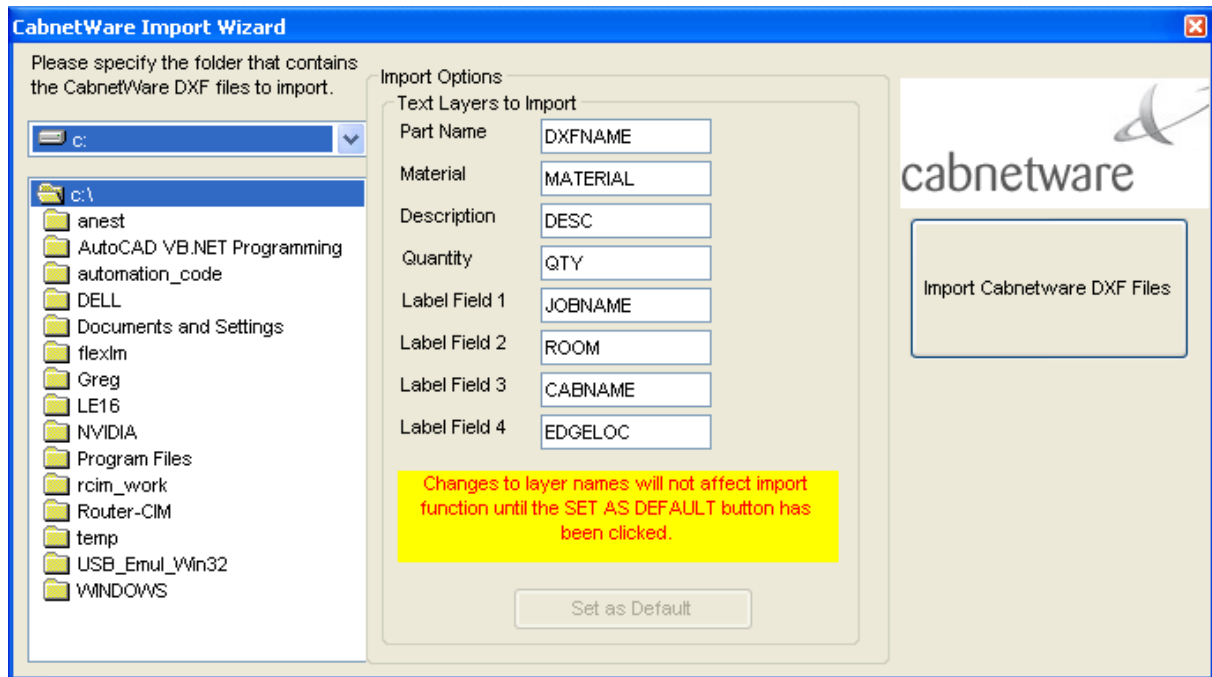
The parts which have DXF names that include an "F" as the fourth character are all entered as two parts automatically by the system, one which has the "F" in the fourth position, and one that has a "B" in the fourth position. When Router-CIM Automation Suite reads them in, it determines that identifying character and sets the job up to cut the "B" part as a separate part from the nested parts. It is expected that the Job Editor will have both parts in it when the import is complete, but the "B" part will not appear in the cncrun.txt file. If you then select the "B" part, you will notice in its properties that it has the flag set to be cut as a single operation.

If the "B" part exists in the Cncrun file, then you will see three parts with similar names, as the system automatically loads the "B" part when the "F" part is present. If it encounters a "B" part in the list, Router-CIM Automation Suite will treat that as another part in the job, which likely will be incorrect.

Cabnetware

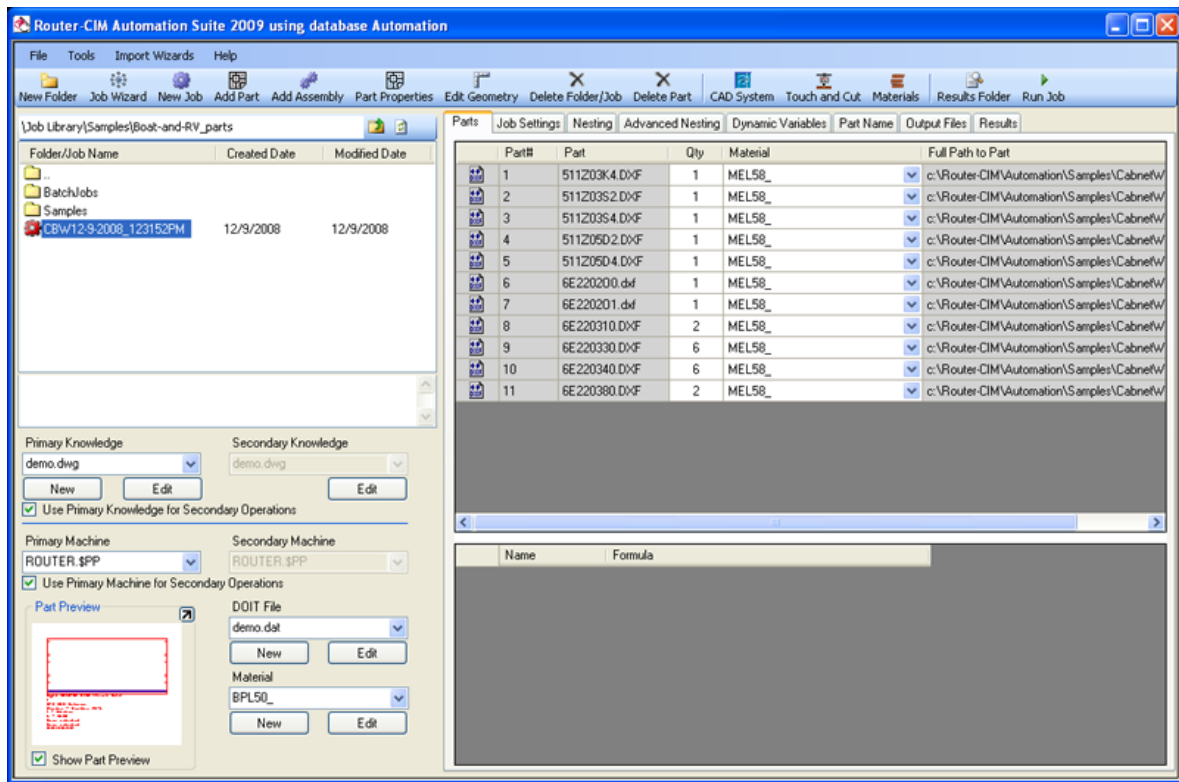
Cabnetware integrates room and cabinet design capabilities with manufacturing tools. Cabnetware is for residential and commercial manufacturers of casework; allowing them to generate detailed drawings for use with building contractors and architects. This program has modules that allow it to export layered DXF files that can be imported into Router-CIM Automation Suite with the help of the Cabnetware import wizard.

Select the Cabnetware wizard from the list and you are presented with the following window.



Select the folder where the DXF files are located on the left side. In the middle column you may edit the layers that the Cabnetware software presents it data on. Normally this is only done once and then left alone, however the option exists to change it should this be necessary.

Once the folder containing the DXF files is selected, press the Import Cabnetware DXF Files button and a job will be added to the current folder.



The Text Layers in Cabnetware DXF files contain information such as Material, Quantity, Job Name, DXF Name, etc... These are needed to process the parts through Router-CIM Automation Suite. The text is typically located in the lower left corner of the DXF.



Right DRAWER SIDE (Front Side)

1
 5/8 G2S Melamine
 Woodco - Boulder FS-A
 1 - Kitchen
 6 - BASE
 None selected
 None selected
 511203K4.DXF

Settings in Cabnetware

The following are the default layer names that should be used in Cabnetware.

- The outside of the part should always be placed on a layer that begins with **BORDER**
- All vertical drill holes, regardless of size, should be placed on layer **VBORE**
- Dadoes and rabbet's with a width of .25 inch should be placed on layer **ROUTE 250**
- Dadoes and rabbet's with a width of .375 inch should be rectangles on layer **ROUTE 375**
- Dadoes and rabbet's with a width of .500 inch should be rectangles on layer **ROUTE 500**
- Dadoes and rabbet's with a width of .625 inch should be rectangles on layer **ROUTE 625**
- Dadoes and rabbet's with a width of .750 inch should be rectangles on layer **ROUTE 750**

The following layers are for the text in each DXF file (Text height should be .125 and Text spacing should be .130).

- Part description should be placed on a layer called **DESC**
- Part quantity should be placed on a layer called **QTY**
- Part material should be placed on a layer called **MATERIAL**
- Job name should be placed on a layer called **JOBNAME**
- Room name should be placed on a layer called **ROOM**
- Cabinet name should be placed on a layer called **CABNAME**
- Edge band location should be placed on a layer called **EDGELOC**
- Edge band material should be placed on a layer called **EDGEMAT**
- DXF filename should be placed on a layer called **DXNAME**

Layer	Knowledge	Tool	Spindle#
Border	Border	.5" Compression	1
Vbore	Vbore	Multi Spindle Gang Drill	Varies
Route250 Route375	Dado250	.25 Router Bit	2
Route500 Route625 Route750	Dado500	.5" Down Shear	3

Table 1. Layer to Knowledge Association for Cabnetware Router-CIM

Cabnetware Notes:

In the Cabnetware "CNCAPP.INI" file the following should be set:

- Reverse Panel Reference = Y
- Reverse Verticals Reference = Y
- Output DXF Rects = Y

To access the "CNCAPP.INI," in the Cabnetware CNC Link, go to "Help" pull down menu and select "About Cabnetware CNC" and then click on the purple disk icon to edit the INI file.

All geometry that is to be cut in like fashion (same tool, spindle, feed rate etc.) should be placed on the same layer.

Typically, dadoes and rabbet's should be output as rectangles that overlap the edge of the material by one tool radius (if the cut is to extend to the edge of the part).

All geometry should be output with negative thickness indicating cut depth.

Backside Parts with Cabnetware

There is a provision in Cabnetware to identify the parts which have backside or secondary operations. Likewise in Router-CIM Automation Suite there is a similar provision.

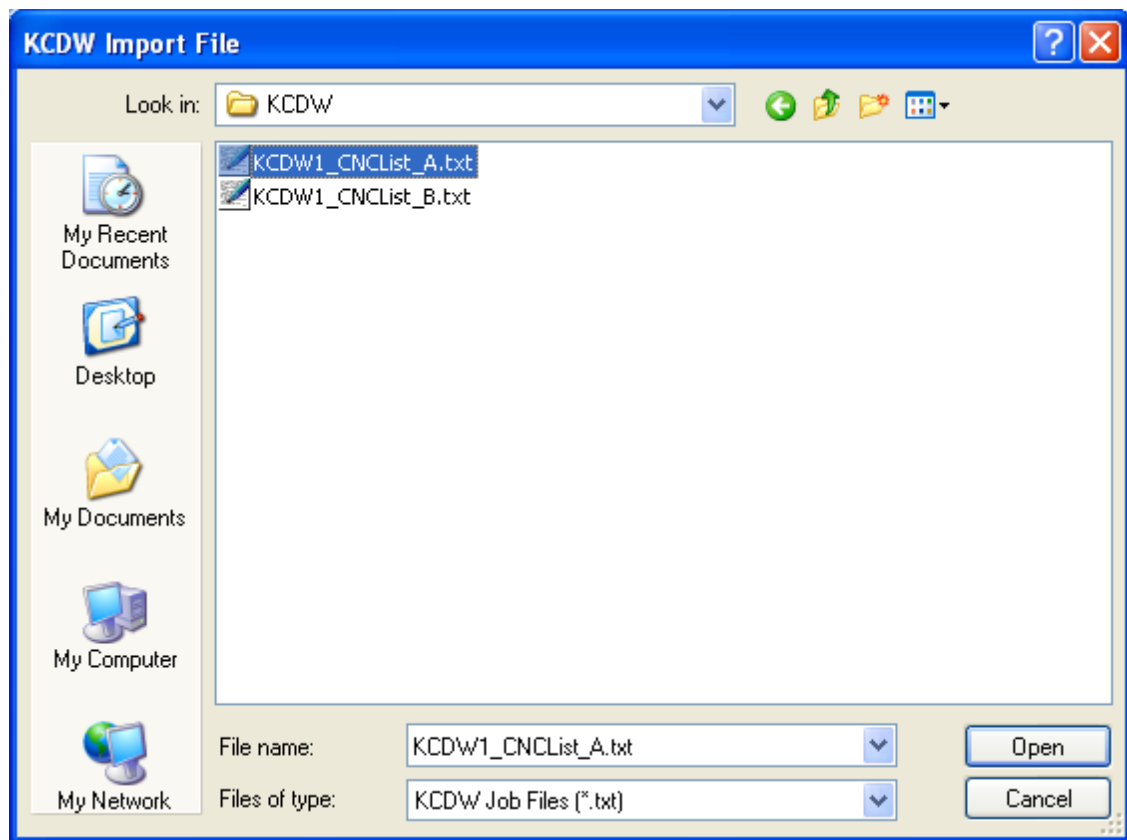
The parts which have DXF names that end in 1, 3, 5 are all treated as backside parts. When Router-CIM Automation Suite reads them in, it determines that last character and sets the job up to cut that as a separate part from the nested parts. It is likely in these types of jobs to have the same part name, except for the last character. The one that ends in zero (0) for instance will be nested and the one that has a 1, 3 or 5 will be cut as a single operation.

KCDw

With the KCDw Machining version, layered DXF files are created and stored in a folder of your choosing. It will also create a job file that describes the material, part names, etc. Once the DXF files are created, select the KCDw Import Wizard in Router-CIM Automation Suite.

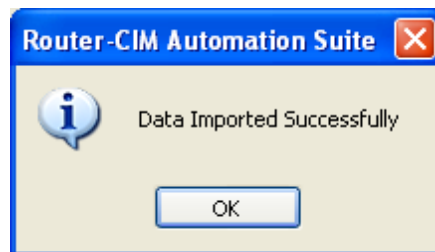


Click on Select Data File and a new window will appear allowing you to select the job file that KCDw created.



There is typically only one job file in each of the folders where the job DXF files are created. Select the job file and the parts will be inserted into Router-CIM Automation Suite as a job in the current folder.

A confirmation window will show up when the file is imported successfully.



Original Excel/Comma Delimited Import Wizard (For exceptions only)

The Original Excel/Comma Delimited Import Wizard (For exceptions only) is a legacy format that allows you to import parts from a file with an excel or comma-delimited format.

Note: In order to use an Excel or Comma Delimited (CSV) to import Assemblies/Jobs instead of parts, please click [here](#) for more information.

The comma delimited import wizard allows you to import parts from a file with a comma-delimited format. This means that there is a comma separating each column of data and each column contains the same type of data. This is very similar to the Excel Spreadsheet format, as the files are very similar.

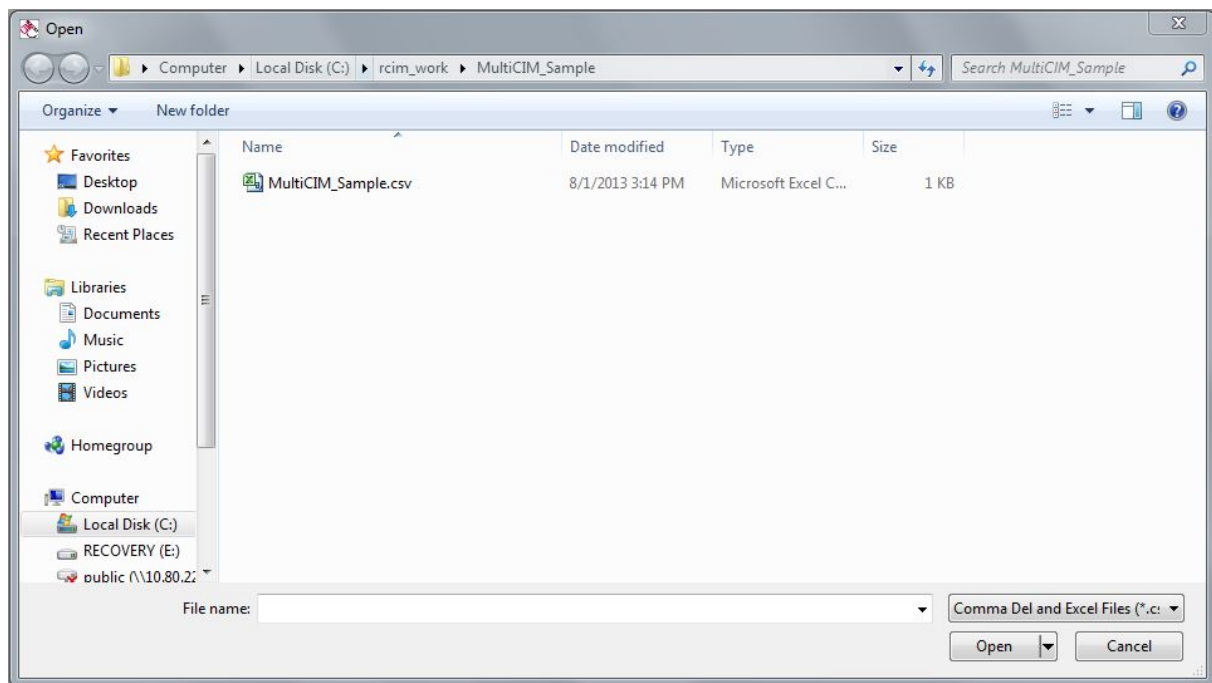
The screenshot shows the 'Comma Del Import Wizard' window. It has a blue title bar and a light gray background. The window is divided into several sections:

- Required Steps:** Contains a 'Select file to import' button, a 'Number of header rows' input field with the value '0', and an 'Assign column headings' button.
- Optional Features:** Contains a 'Select a saved format' dropdown menu, a 'Part Material Field Selection' section with radio buttons for 'Material Code' and 'Material Description' (the latter is selected), a 'Save format (header rows and column headings)' button, a 'Delete selected format' button, and a 'Refresh Data' button.
- Field Selection:** A table with 8 columns labeled 'Field 1' through 'Field 8'. Each column has a 'Select Data T...' dropdown menu.
- Status:** A section at the bottom left with a 'Status:' label and a 'Data File:' input field.

The 'Import' button is located in the top right corner of the window.

Select File to Import

When you select the Original Excel/Comma Delimited Import Wizard (For exceptions only) Import Wizard, a dialog box will appear where you can select an Excel or .CSV file to import into Router-CIM Automation Suite.



Assign Column Headings

Assigning Column Headings will allow you to select the data for each column in the imported file. The choices available in the pulldown list are:

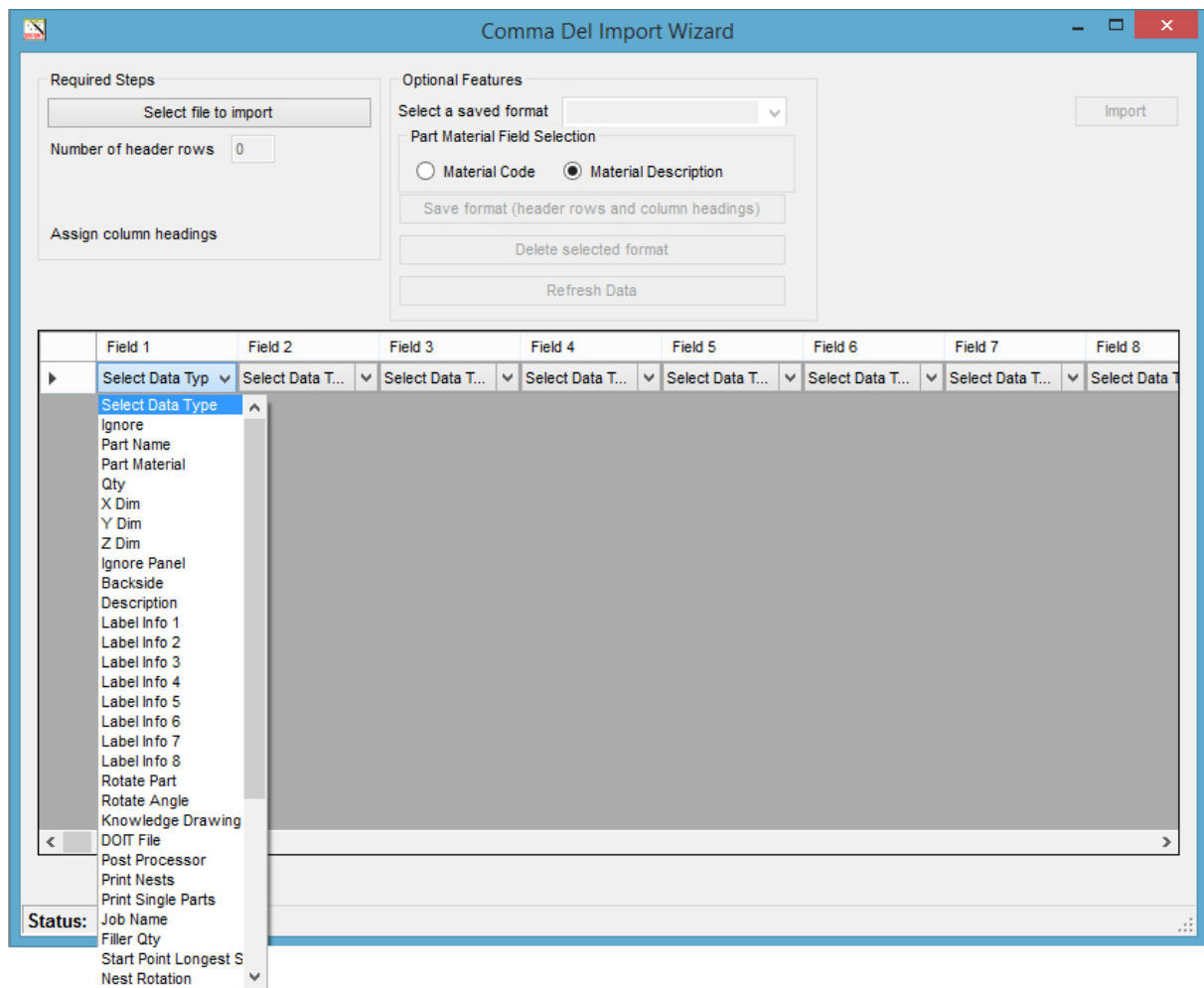
Note: Part Name, Part Material and Quantity must be assigned to a column in order for Router-CIM Automation Suite to import a file.

Variable	Field Option	Description	Value Accepted
Ignore	Optional	Ignores entire column during the import process	N/A
Part Name	Required	The 'Part Name' variable will define the location of the part on the computer. This field should be reserved for the file path location.	File Path Location
Part Material	Required	This field is used for defining the material of the part. For a seamless transition through the RCIM import wizard, the material used should match the material description in the RCIM Materials Database.	Material Description
Quantity (Qty)	Required	This field is used for defining the number of parts needed.	Whole Number
X Dim	Macros Only	Defines the XDIM variable with use in a parametric macro	Real Number, 30 Character Maximum
Y Dim	Macros Only	Defines the YDIM variable with use in a parametric macro	Real Number, 30 Character

			Maximum
Z Dim	Macros Only	Defines the ZDIM variable with use in a parametric macro	Real Number, 30 Character Maximum
Ignore Panel	Macros Only	This variable controls if the 'Panel' layer of a parametric macro is ignored during the automation process	0 (No) or 1 (Yes)
Backside	Macros Only	This variable controls if an individual parametric macro has an associated backside macro	0 (No) or 1 (Yes)
Description	Optional	This variable controls the information included under the 'Record Desc' field located in the part properties window. Each part can be imported with a unique description for labeling purposes	80 Character Maximum
Label Info 1	Optional	This variable controls the information included under the 'Label Field 1' located in the part properties window. Each part can be imported with unique label information.	25 Character Maximum
Label Info 2	Optional	This variable controls the information included under the 'Label Field 2' located in the part properties window. Each part can be imported with unique label information.	25 Character Maximum
Label Info 3	Optional	This variable controls the information included under the 'Label Field 3' located in the part properties window. Each part can be imported with unique label information.	25 Character Maximum
Label Info 4	Optional	This variable controls the information included under the 'Label Field 4' located in the part properties window. Each part can be imported with unique label information.	25 Character Maximum
Label Info 5	Optional	This variable controls the information included under the 'Label Field 5' located in the part properties window. Each part can be imported with unique label information.	99 Character Maximum
Label Info 6	Optional	This variable controls the information included under the 'Label Field 6' located in the part properties window. Each part can be imported with unique label information.	99 Character Maximum
Label Info 7	Optional	This variable controls the information included under the 'Label Field 7' located in the part properties window. Each part can be imported with unique label information.	99 Character Maximum

Label Info 8	Optional	This variable controls the information included under the 'Label Field 8' located in the part properties window. Each part can be imported with unique label information.	99 Character Maximum
Rotate Part	Optional	This variable controls the part option to rotate the individual part. The rotation of the part occurs once the job has started to be processed and prior to the processing of the actual part itself.	0 (No) or 1 (Yes)
Rotate Angle	Optional	This variable controls the amount of rotation (1 degree increments) when the Rotate Part option is selected	Whole Numbers, 0 – 360
Knowledge Drawing	Optional	This variable defines the 'Knowledge Drawing' that will be used to process the part. This option can be different per part when 'Code As Single Part Only' is selected in the part properties window. This variable must be the same as the other parts within a job when nesting.	30 Character Maximum (must include .dwg)
DOIT File	Optional	This variable defines the 'DOIT File' that will be used to process the part. This option can be different per part when 'Code As Single Part Only' is selected in the part properties window. This variable must be the same as the other parts within a job when nesting.	30 Character Maximum (Must include .dat)
Post Processor	Optional	This variable defines the 'Post Processor' that will be used to process the part. This option can be different per part when 'Code As Single Part Only' is selected in the part properties window. This variable must be the same as the other parts within a job when nesting.	30 Character Maximum (must include .\$pp)
Print Nests	Optional	This variable controls the option for printing the created nested sheet after a job has processed.	0 (No) or 1 (Yes)
Print Single Parts	Optional	This variable controls the option for printing the single part after a job has processed when 'Code as Single Part Only' is selected.	0 (No) or 1 (Yes)
Job Name	Optional	This variable controls the name of the job created in Router-CIM Automation Suite	30 Character Maximum. NO SPACES.
Filler Quantity (Filler Qty)	Optional	This variable controls the amount of filler parts that are available to nesting per the particular part	4 Numeric Character Maximum or -1
Start Point Longest Side	Optional	This variable controls the option to have the start point of the cut cycle be situated on the longest side of a particular piece of geometry for the particular part	0 (No) or 1 (Yes)

Nest Rotation	Optional	This variable controls the rotation of the part within the material that it will be nested with. The part and the material can have defined nest rotation variables that allow for nesting to handle the part different than what is defined on the nesting material	Same As Material; ALL; 0; 0 90; 0 90 180
Mirror	Optional	This variable controls whether a part will be the mirror image. Direction of the mirror is controlled by the variable 'Mirror Type'.	0 (No) or 1 (Yes)
Mirror Type	Optional	Defines the axis on which the part will be mirrored	Horizontal or Vertical
Variable	Optional	User defined fields for additional parametric macro variables	
Packing Direction	Optional	Defines the direction that the nesting engine will fill up the sheet with the parts defined by the job.	3 (Auto) 4 (Horizontal) 5 (Vertical)
Nesting Start	Optional	Defines the corner that the nesting engine will start to fill up the sheet with the parts defined by the job.	1 (Lower Left) 2 (Upper Left) 3 (Upper Right) 4 (Lower Right)
Sheet Origin	Optional	Defines the corner of the sheet that the output code files will base the X and Y axis direction on. Ex. 1 would tell the code to be based off the lower left of the sheet meaning positive X and positive Y.	1 (Lower Left) 2 (Upper Left) 3 (Upper Right) 4 (Lower Right)
NC Code Extension	Optional	This will be the file extension that the output files will be created with. The extension is the only thing needed. You do not have to define the decimal point. Ex. TXT, CNC, OUT, etc.	Any 2 or 3 digit file extension that your CNC accepts
Variable	Optional	User defined fields for additional parametric macro variables	



If you already have an import format saved, you can select it from the drop-down at this time:

Comma Del Import Wizard

Required Steps

Select file to import

Number of header rows: 0

Assign column headings

Optional Features

Select a saved format: [Dropdown]

Part Material Field Selection: SOLID-CIM3D_2016

☐ Material Code ☒ Material Description

Save format (header rows and column headings)

Delete selected format

Refresh Data

Import

	Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9
▶	Select Dat...	Select Dat...	Select Dat...	Select Dat...	Select Dat...	Select Dat...	Select Dat...	Select Dat...	Select D...
	C:\rcim_workl...	PLEX_0.2500	2	36.9375 x 2.43...		36.9375	2.4375	0.2500	0
	C:\rcim_workl...	PLYWD_0.7500	1	42.5000 x 3.00...		42.5000	3.0000	0.7500	1
	C:\rcim_workl...	MDF_0.5000	1	60.0000 x 27.2...		60.0000	27.2500	0.5000	1
	C:\rcim_workl...	PLYWD_0.7500	1	60.0000 x 8.00...		60.0000	8.0000	0.7500	1
	C:\rcim_workl...	PLYWD_0.5000	1	37.6450 x 22.4...		37.6450	22.4850	0.5000	1
	C:\rcim_workl...	PLEX_0.2500	1	59.5000 x 8.50...		59.5000	8.5000	0.2500	1
	C:\rcim_workl...	PLYWD_0.2500	1	58.0000 x 7.50...		58.0000	7.5000	0.2500	0
	C:\rcim_workl...	MDF_0.5000	1	60.0000 x 27.2...		60.0000	27.2500	0.5000	0
	C:\rcim_workl...	PLYWD_0.7500	1	59.5000 x 8.50...		59.5000	8.5000	0.7500	1
	C:\rcim_workl...	PLYWD_0.7500	2	25.7500 x 8.00...		25.7500	8.0000	0.7500	1
	C:\rcim_workl...	PLYWD_0.7500	1	60.0000 x 8.00...		60.0000	8.0000	0.7500	1
	C:\rcim_workl...	PLEX_0.3750	1	56.7500 x 22.5...		56.7500	22.5000	0.3750	1

Status: Idle Data File: C:\rcim_work\SCIM\

Save Format (header rows and column headings)

You are able to save the data that is set up for a job by selecting '**Save Format (header rows and column headings)**'. You will be prompted for a name of the Saved Format:

Saved Format Name

Please enter a name for this format.

OK

Cancel

User_Import_Format

Selecting OK will add this name to the Saved Format list so that you can select this format at a later time instead of re-selecting each column.

Delete Selected Format

You may delete the saved selected format from the saved format list.

Part Material Field Selection

This option allows you to select whether you want Router-CIM Automation Suite to match the material in your excel or comma delimited file based on the material description or material code that you have set up in the Router-CIM Automation Suite Materials database.

Clear Current Column Format

If you select a saved format but need to start from the default column headings, select the '**Clear Current Column Format**' button to set the column headings to the default.

Number of Header Rows

If your comma delimited file has header rows in it for any reason, you can bypass those and get straight to the data by specifying how many header rows to skip.

Refresh Data

This option refreshes the screen, displaying all column headings selected or imported.

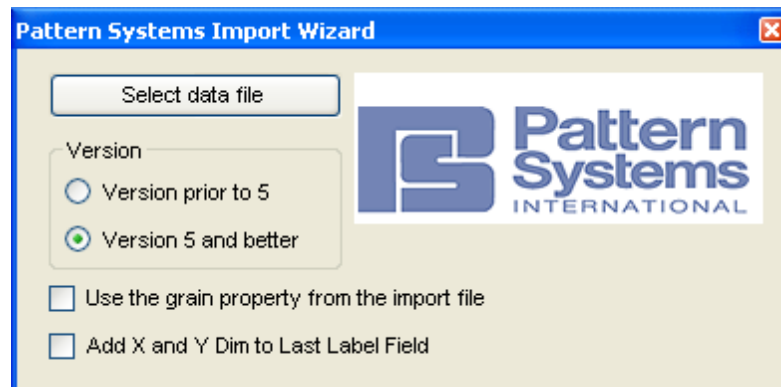
Import

Finally when you are done selecting your file and headings, you can import the file into Router-CIM Automation Suite and it will build a job from the selected data

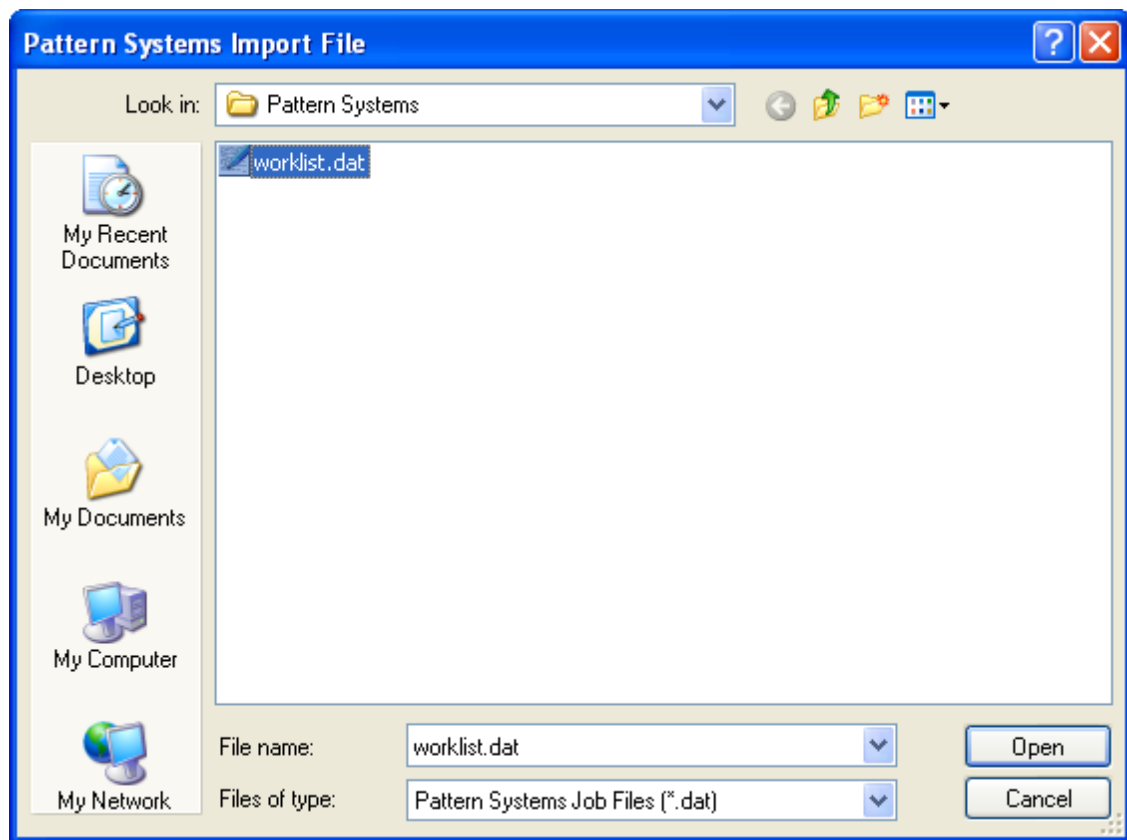
Pattern Systems

Pattern Systems software allows the creation of layered DXF files for import. Once the DXF Files have been made, Pattern Systems will also create a Job File that describes the material used, the part names, quantities, edge treatments, etc. That Job file can be selected and Router-CIM Automation Suite will import the files into a Router-CIM Automation Suite job.

Select the Pattern Systems Wizard and a window will appear allowing you to select the data file from Pattern Systems. Click Select Data File.



A new window will appear allowing you to navigate to the folder where the job and DXF files you made with Pattern Systems are stored.



Select the data file and click on Open. The files will be imported into Router-CIM Automation Suite as a job in the current folder.

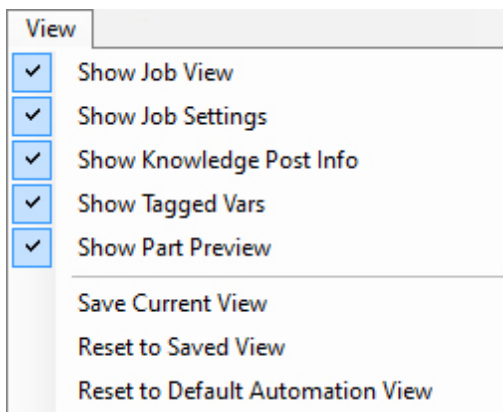
View

The following areas can be moved, floating or docked to customize the user interface to your liking:

- 1) Job Tree
- 2) Job Settings
- 3) Knowledge Post Info
- 4) Tagged Vars
- 5) Preview

You can also turn areas on/off by selecting the ones you want under the 'View' options.

- Show Job Tree
- Show Job Settings
- Show Knowledge Post Info
- Show Tagged Vars
- Show Part Preview



Save Current View

If you make adjustments to the Router-CIM Automation Suite interface, you can save that layout by selecting the 'Save Current View' from the 'View' dropdown.

Reset to Saved View

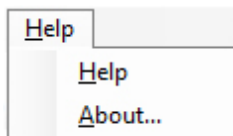
If you make adjustments to your Router-CIM Automation Suite interface and you would like to return to your saved view, select the 'Reset to Saved View' from the 'View' dropdown.

Reset to Default Automation View

There are moveable windows in Router-CIM Automation Suite. When selecting this option, it will reset the view to the default Router-CIM Automation Suite view.

For more information on moving the interface windows, refer to the ['Moving Windows in Automation'](#) section.

Help Menu



Help

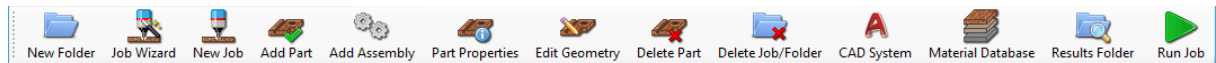
Selecting 'Help' opens the offline Help file.

About

About shows an informational window displaying the current version of your Router-CIM Automation Suite software.

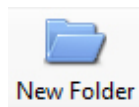


Toolbar

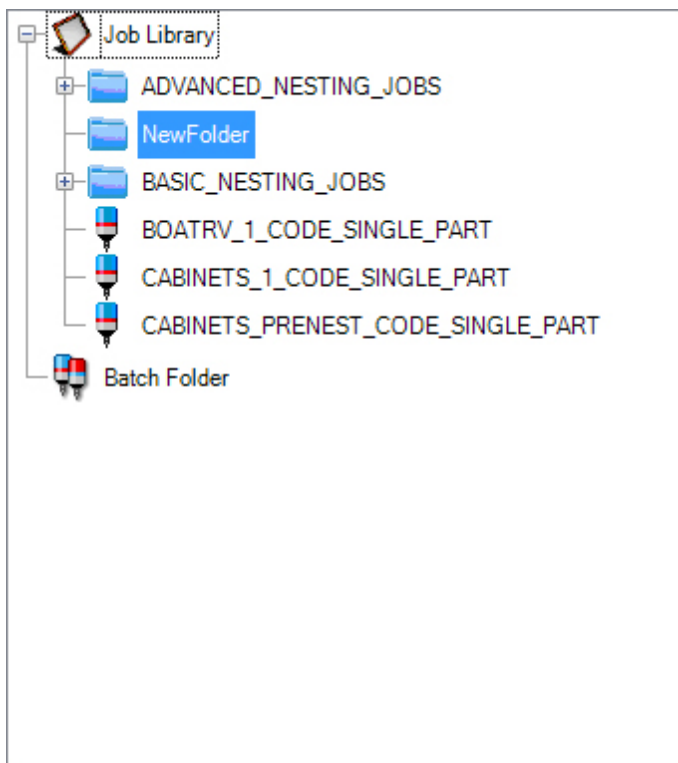


The toolbar in Router-CIM Automation Suite allows access to many functions within jobs, parts, and folders.

New Folder



Using this option will create a new folder in the current position of the job tree.

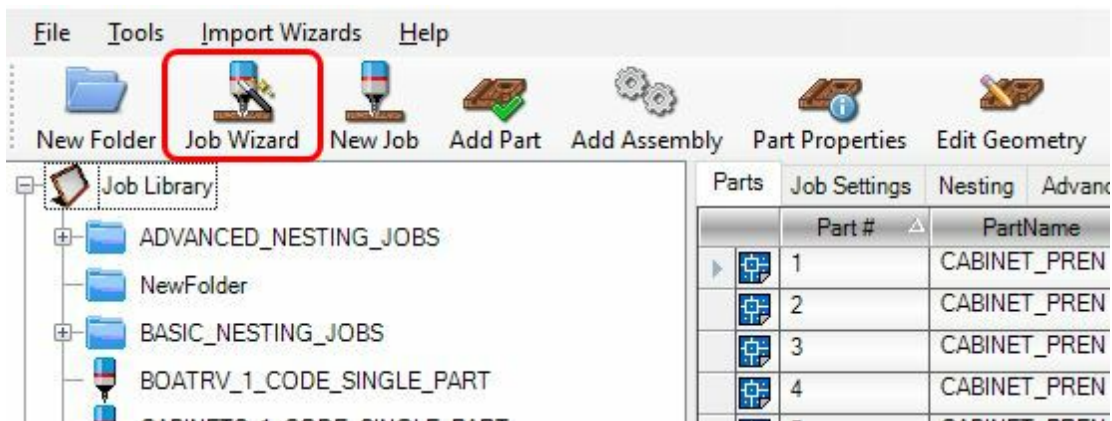


Job Wizard



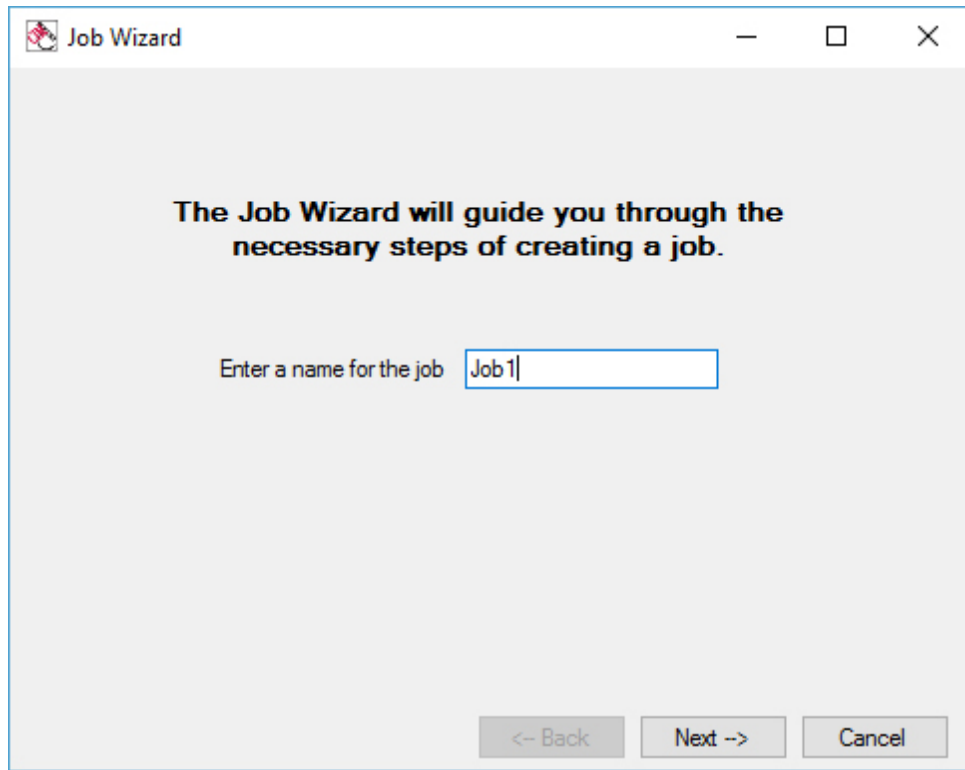
Using the Job Wizard allows you to create a job in a step by step fashion.

Select 'Job Wizard'



Enter a Job Name and then select 'Next'

The job name will also be the output folder name where the job results are stored.



The Job Wizard will guide you through the necessary steps of creating a job.

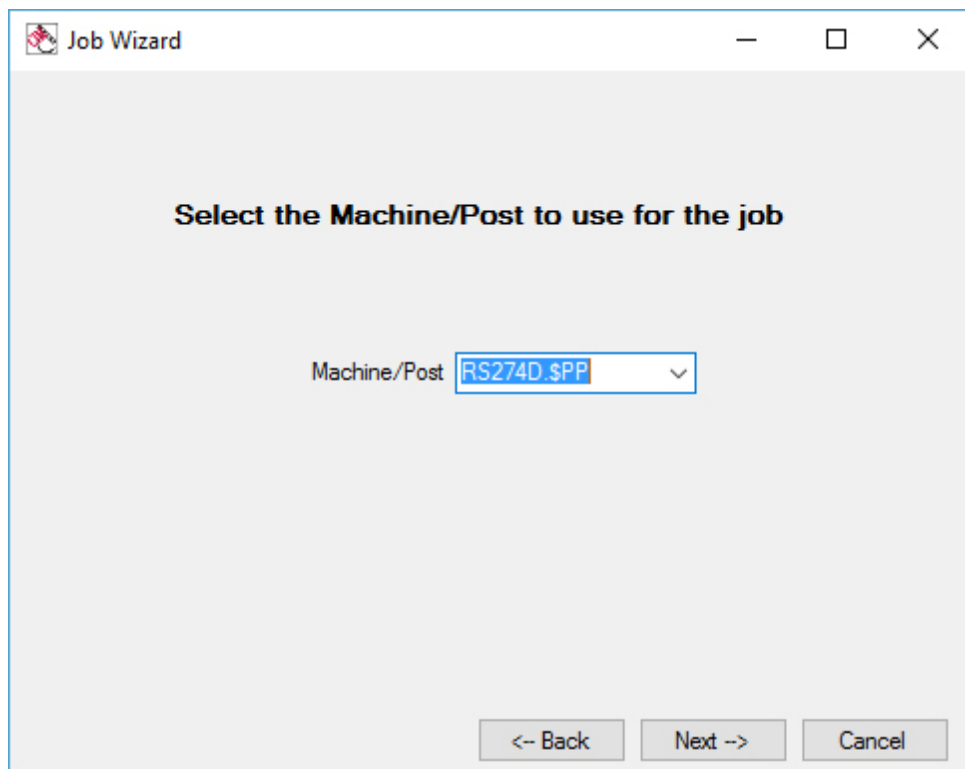
Enter a name for the job

<-- Back Next --> Cancel

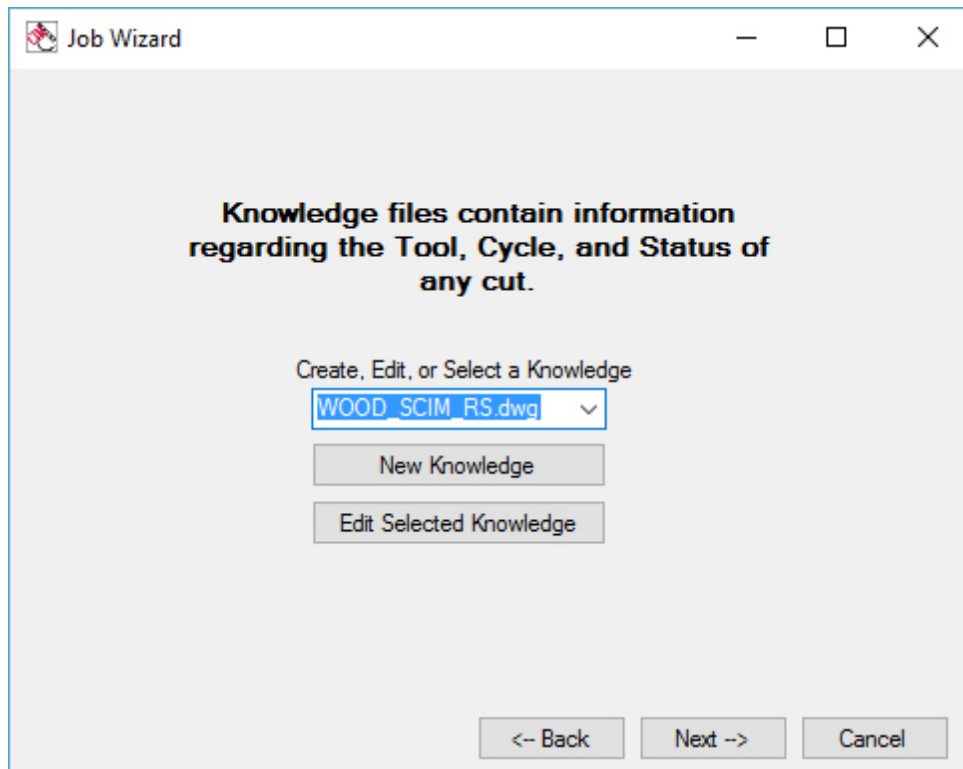
The image shows a 'Job Wizard' dialog box with a title bar containing a red icon and the text 'Job Wizard'. The main area has a light gray background with the text 'The Job Wizard will guide you through the necessary steps of creating a job.' in bold. Below this is a text input field with the label 'Enter a name for the job' and the text 'Job1' inside. At the bottom right are three buttons: '<-- Back', 'Next -->', and 'Cancel'.

Select your Machine/Post processor and then select 'Next'

The post processor makes the machine code for different types and configurations of CNC machines.

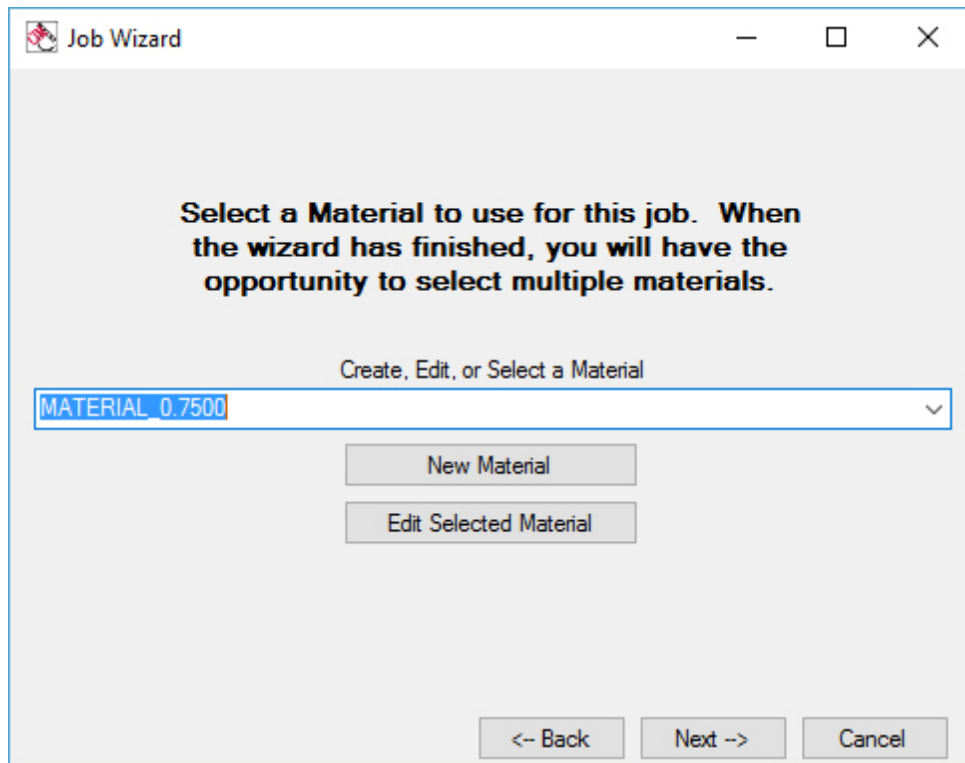
**Create, Edit or Select a Knowledge drawing and then select 'Next'**

The knowledge drawing is a collection of machining operations that are saved and named within this drawing. You can make a new knowledge drawing where you build knowledges and save /name them. You then save that drawing and use it as your knowledge drawing. From this screen, you can also edit existing knowledges that are stored in a selected knowledge drawing.



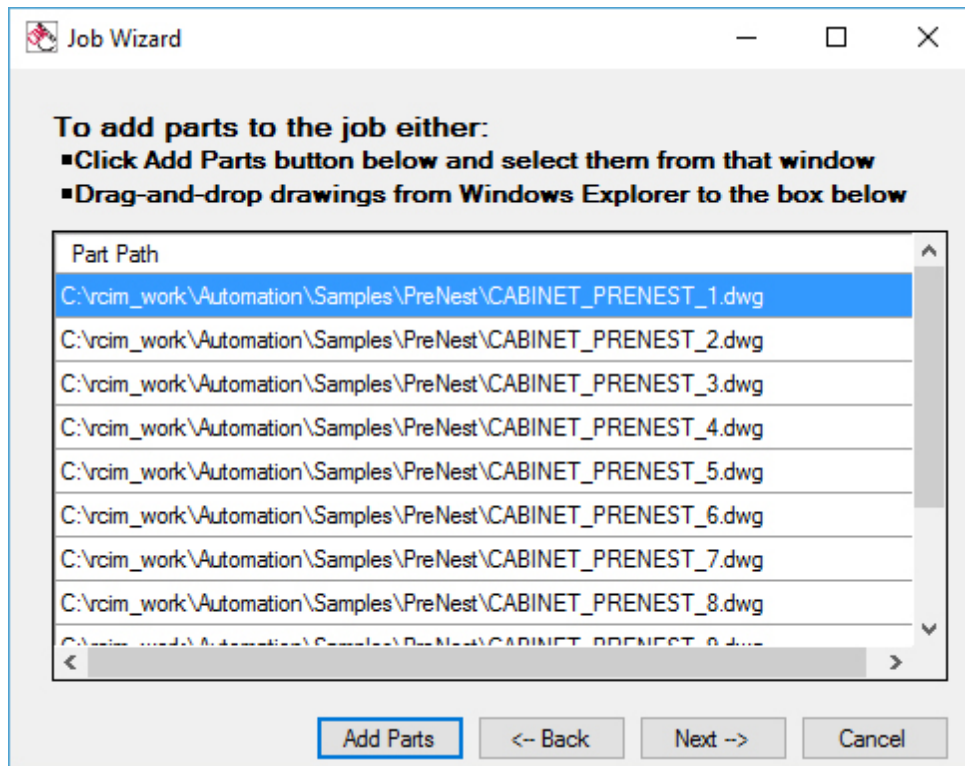
Create, Edit, or Select a default material and then select 'Next'

The material determines the sheet size when nesting parts. The material also determines the material thickness and if Z zero is the top of the material or top of spoil board.



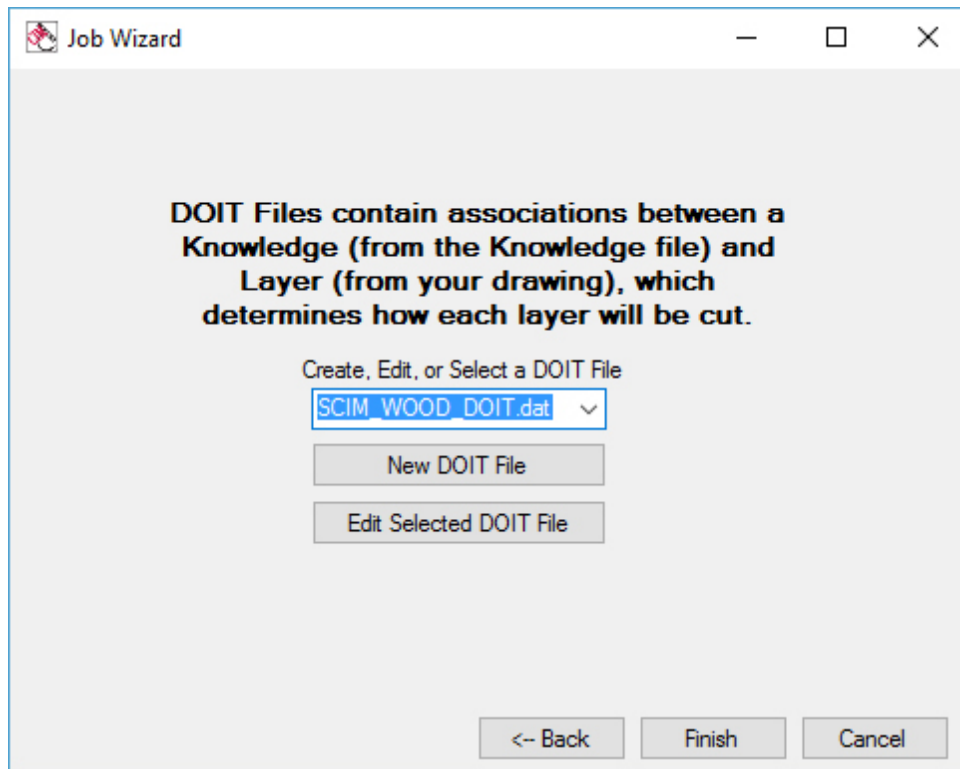
Add parts to your job and then select 'Next'

From this screen, you can add parts to your Router-CIM Automation Suite job. Add parts by picking the Add parts button and browsing for DWG, DXF, or SCN files to add. You can also drag files from Windows Explorer and drop them into this window.

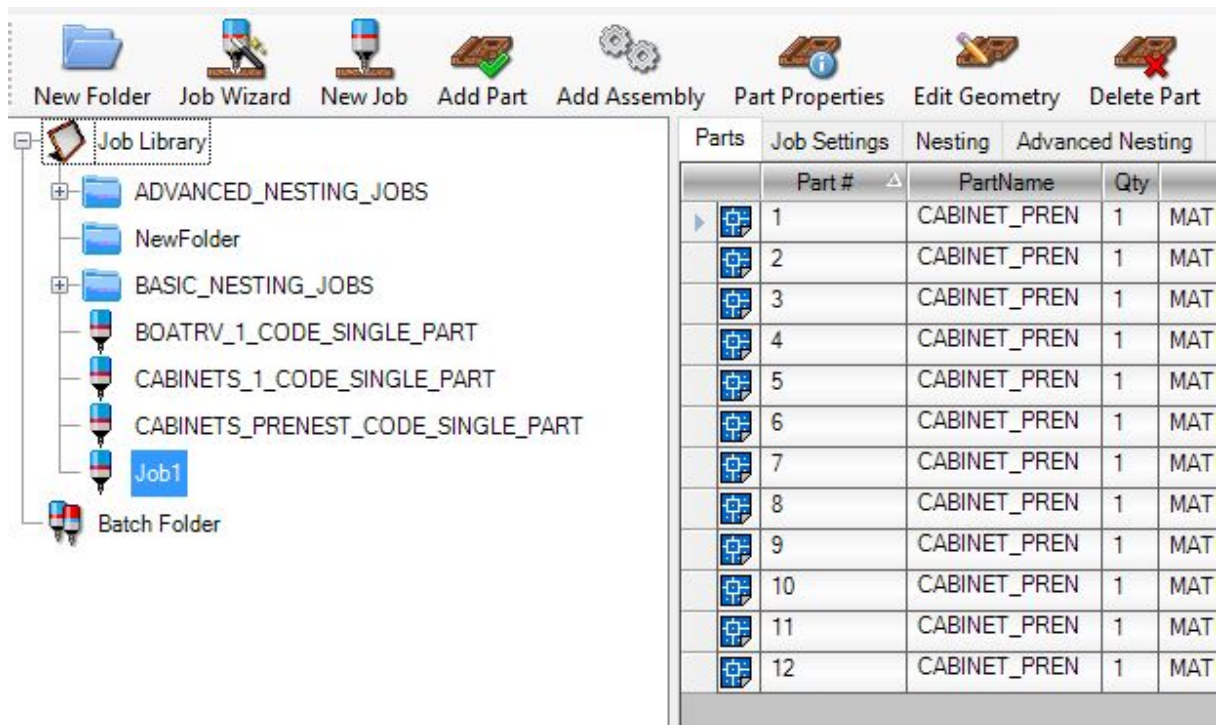


Create, Edit, or Select a DOIT file and then select 'Finish'

The DOIT file contains the association list that cuts a specific layer with a specific knowledge. You can make a new DOIT file and choose what knowledge cuts what layer or you can edit the selected DOIT file.



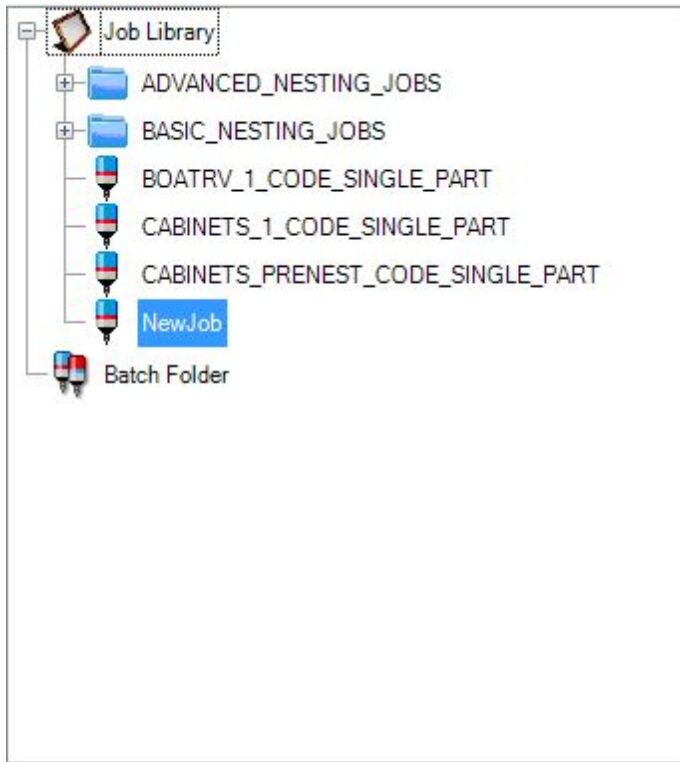
Once you select **'Finish'**, Router-CIM Automation Suite will build a job based on the information you have provided.



New Job

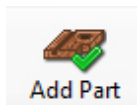


This option will create a new job in the currently select folder.



The new job will have the name of 'NewJob' until you rename it. The job will contain no parts, but will have all the default settings in use.

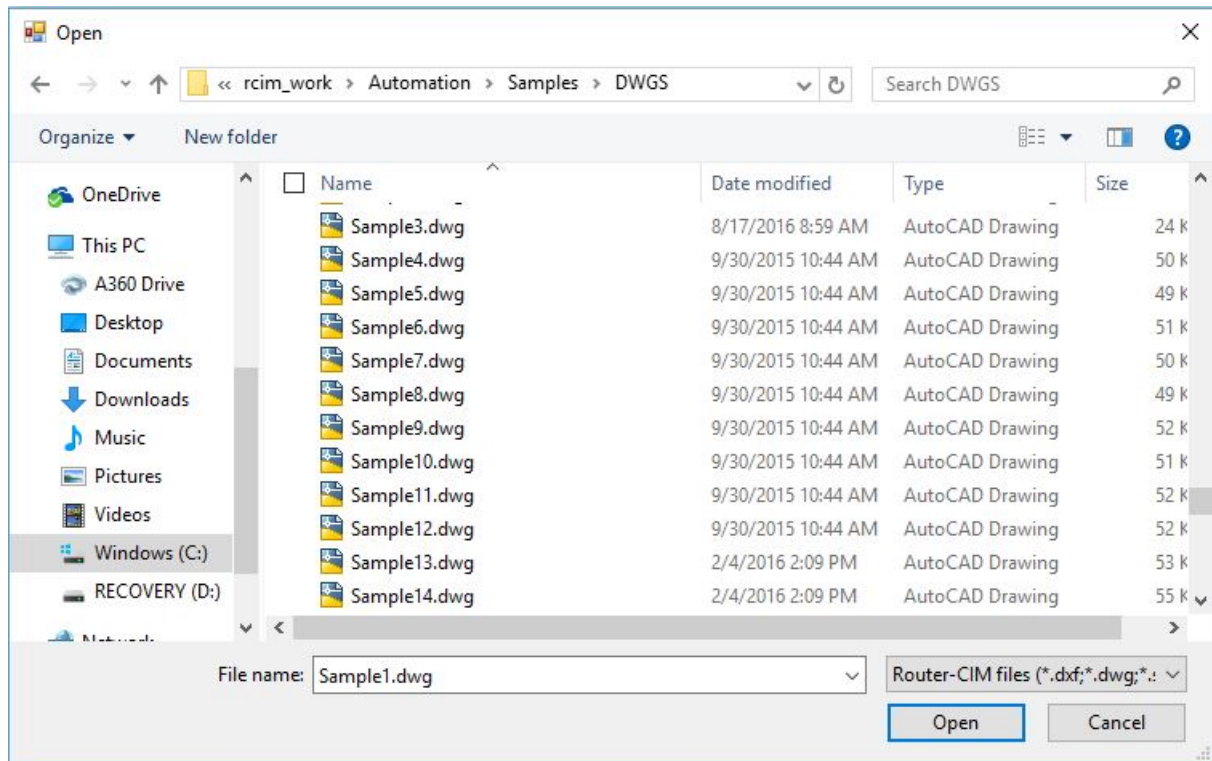
Add Part



Selecting this option will open the Add Parts window. This window has several options, you can select default folders rcim_work, or Router-CIM as well as set the default types of files to DWG, DXF, SCN or MPR files. You can see a preview of the part, unless multiple parts are selected.

Note: MPR files are converted from the original format and re-saved as a DWG format within the folder you selected the MPR file from. For more information, go to the ['MPR to DWG Converter'](#) section.

Note: Part locations have a 256 character limit.



To scale your parts from inch to metric or metric to inch prior to processing the job, please go to ['Scale Part Geometry in Router-CIM Automation Suite'](#).

MPR to DWG Converter

When an MPR file is added to Router-CIM the MPR file will be converted to a drawing (DWG) file in the background, and you will see the drawing file added to the Router-CIM job. The drawing file will contain geometry that has thickness and is layer separated.

If the material that was specified in the MPR is found in the material database, then it will be used. If a MPR specifies a material that was not found in the database then the material will be put on the default material that was set for the job.

MPR to Drawing Layer Name Format:

Features found inside of the MPR file will be put on unique layers depending on the information found in the MPR file.

Contour features follow the format: "Feature_**Tool**_RadiusCompensation_**Depth**

Tool is the Tool Number

RadiusCompensation will be either Center, Left, or Right based on the parameters NOWRK, WRKL, and WRKR

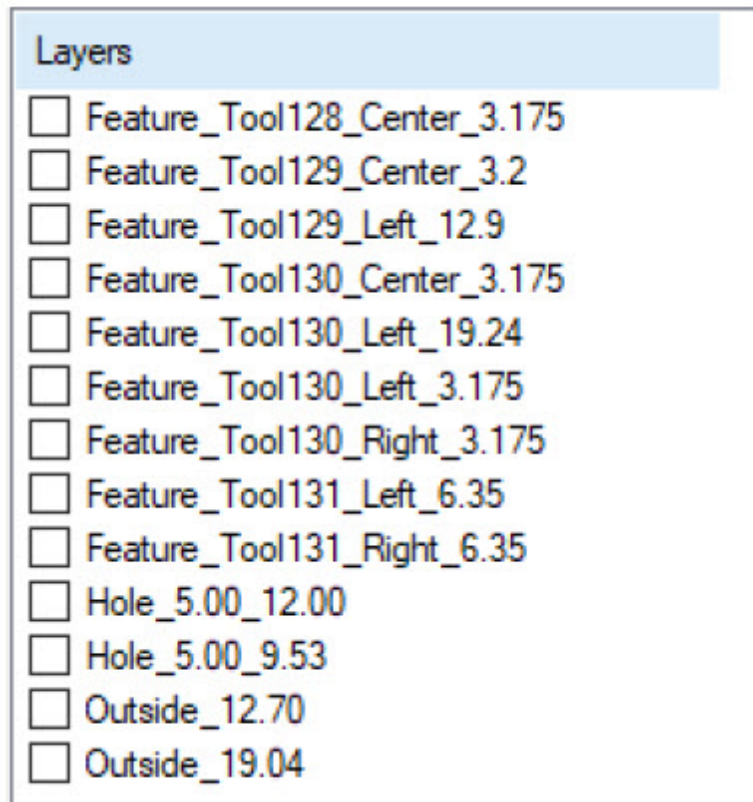
Depth is calculated based on the Start Coordinate Z

Drill features follow the format: "Hole_**Diameter**_**Depth**

Both **Diameter** and **Depth** are generated from the tool found in the MPR file

Outside features will follow the format: "Outside_**Depth**
Depth is generated from the MPR file

Below is a screen capture from the DOIT editor for MPR files that were converted to drawing files.



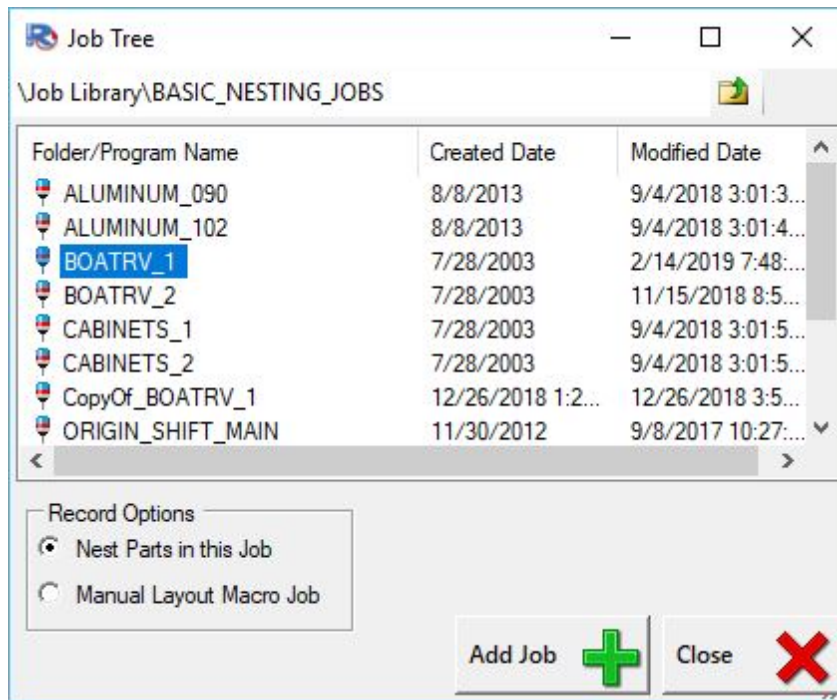
Add Assembly



An assembly is simply another job that already has parts and settings. If you make a new job, and then select 'Add Assembly', you will be prompted with a window to pick the job you would like to add to the current job you have selected.

Note: When adding an assembly, the assembly that was added will be looked at in order to determine the job settings that were used such as Knowledge file, DOIT file, Post Processor, etc.

From the job tree in this window, select the job you want to add:



Then select the job that you want to add and then click on **'Add Job'**.

Select the **'Exit'** button to finish.

The job will be added to the current job as a part.

	Part#	Part	Qty	Material	Full Path to Part
	1	BOATRV_1	1	1/2 Baltic Birch Plywood	BOATRV_1

Part Naming Convention when Assemblies are added to a Job:

When multiple assemblies are added to a job, the processed jobs create a specific part naming pattern.

The pattern is as follows for the parts that are included in the assemblies:

Part 1 of Assembly 1 and the first part in the entire job with Assembly 1 parts continued until all parts in Assembly 1 are named.

Part 1 of Assembly 2 will then follow with a sequential number ordering until all parts in Assembly 2 are named

Part 1 of Assembly 3 will then follow with a sequential number ordering until all parts in Assembly 3 are named

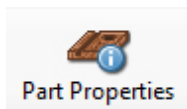
And the pattern continues.

Here is an example for further explanation:

Assembly 1 in the Job has two parts. Assembly 2 in the Job has two parts. Resulting names are as follows

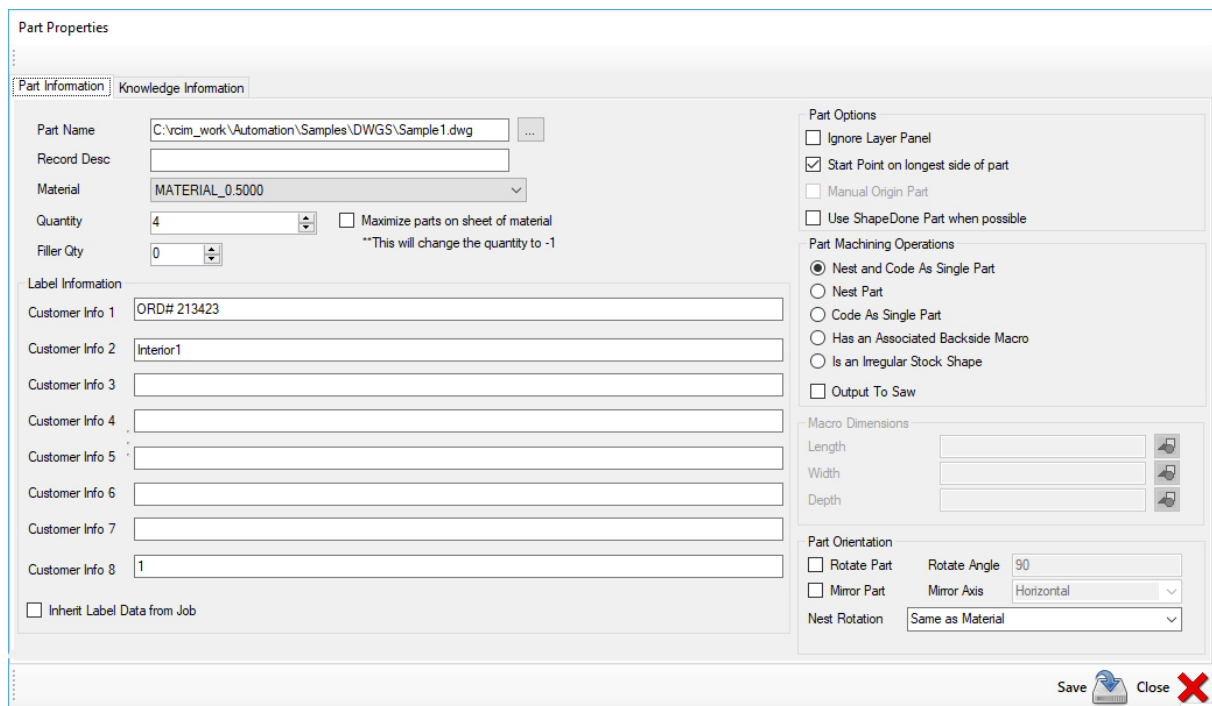
Assembly 1, Part 1 is **1_1** (part 1 of Assembly 1 and first (1) part in the entire Job)
Assembly 1, Part 2 is **2_2** (part 2 of Assembly 1 and second (2) part in the entire Job)
Assembly 2, Part 1 is **1_3** (part 1 of Assembly 2 and third (3) part in the entire Job)
Assembly 2, Part 2 is **2_4** (part 2 of Assembly 2 and fourth (4) part in the entire Job)

Part Properties



This selection opens the part properties window.

For more information on the Part Properties Settings, please go to the ['Part Properties'](#) section located in the Part Window section.



For users that have the Advanced Nesting Option, you will also see the Veneer Matching Parameters:

Part Properties

Part Information Knowledge Information

Part Name: C:\cim_work\Automation\Samples\DWGS\Sample1.dwg

Record Desc:

Material: MATERIAL_0.5000

Quantity: 4 ☐ Maximize parts on sheet of material

Filler Qty: 0 **This will change the quantity to -1

Label Information

Customer Info 1: ORD# 213423

Customer Info 2: Interior1

Customer Info 3:

Customer Info 4:

Customer Info 5:

Customer Info 6:

Customer Info 7:

Customer Info 8: 1

☐ Inherit Label Data from Job

Veneer Match Parameters

Name:

Location Point:

Rotation:

Part Options

☐ Ignore Layer Panel

☒ Start Point on longest side of part

☐ Manual Origin Part

☐ Use ShapeDone Part when possible

Part Machining Operations

☒ Nest and Code As Single Part

☐ Nest Part

☐ Code As Single Part

☐ Has an Associated Backside Macro

☐ Is an Irregular Stock Shape

☐ Output To Saw

Macro Dimensions

Length:

Width:

Depth:

Part Orientation

☐ Rotate Part Rotate Angle: 90

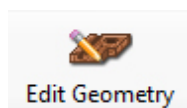
☐ Mirror Part Mirror Axis: Horizontal

Nest Rotation: Same as Material

Save Close

This will show the current properties of the currently selected part in a job. If no part is selected, the first part in the job is shown.

Edit Geometry



Selecting Edit Geometry will open AutoCAD and show the currently selected part opened from its original location. If no part is selected, you will be prompted to select a part first.

Delete Job/Folder



This option deletes a job if a job is selected or an entire folder if a folder is selected. You cannot get these jobs or folders back once they are deleted, so be sure to have a pack and go of the job or a backup of the database if you think you might want them back at some point.

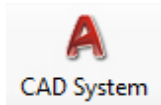
Delete Part



Selecting this option will delete the selected part from within a job.

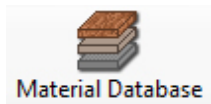
You may also delete a part by right clicking on it and selecting delete from the right-click menu.

CAD System



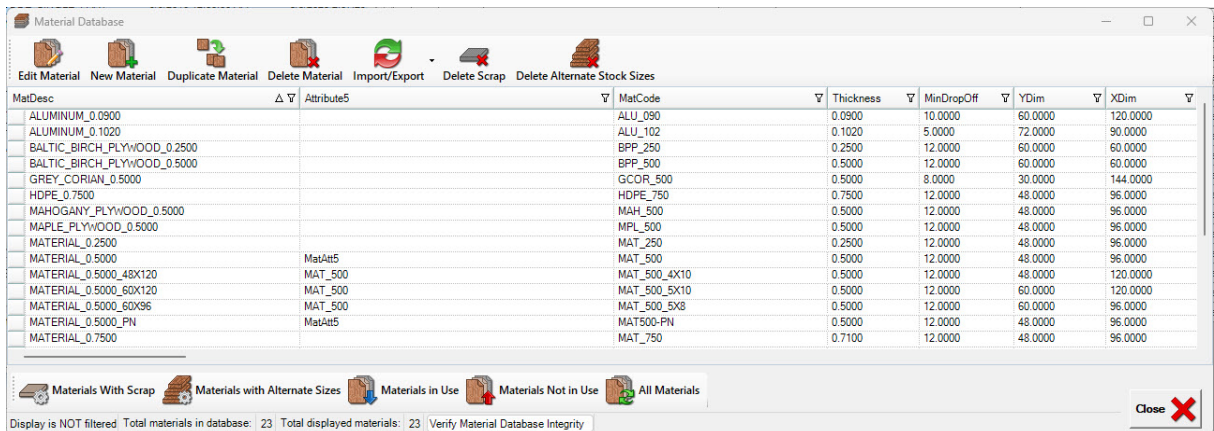
This option is a shortcut to starting AutoCAD. Using this option will start AutoCAD in a new blank or default drawing.

Material Database



The 'Material Database' button opens the Material Database for viewing or editing.

For more information on the Materials Database, please refer to the [Materials Database/Properties](#) section.



Results Folder



Selecting this option will show you the result folder for a job, if a job is selected. If no job is selected, then the base results folder (the one containing all the others) is shown. The default results folder can be specified in the system settings under System Folders >> Data Output Folder.

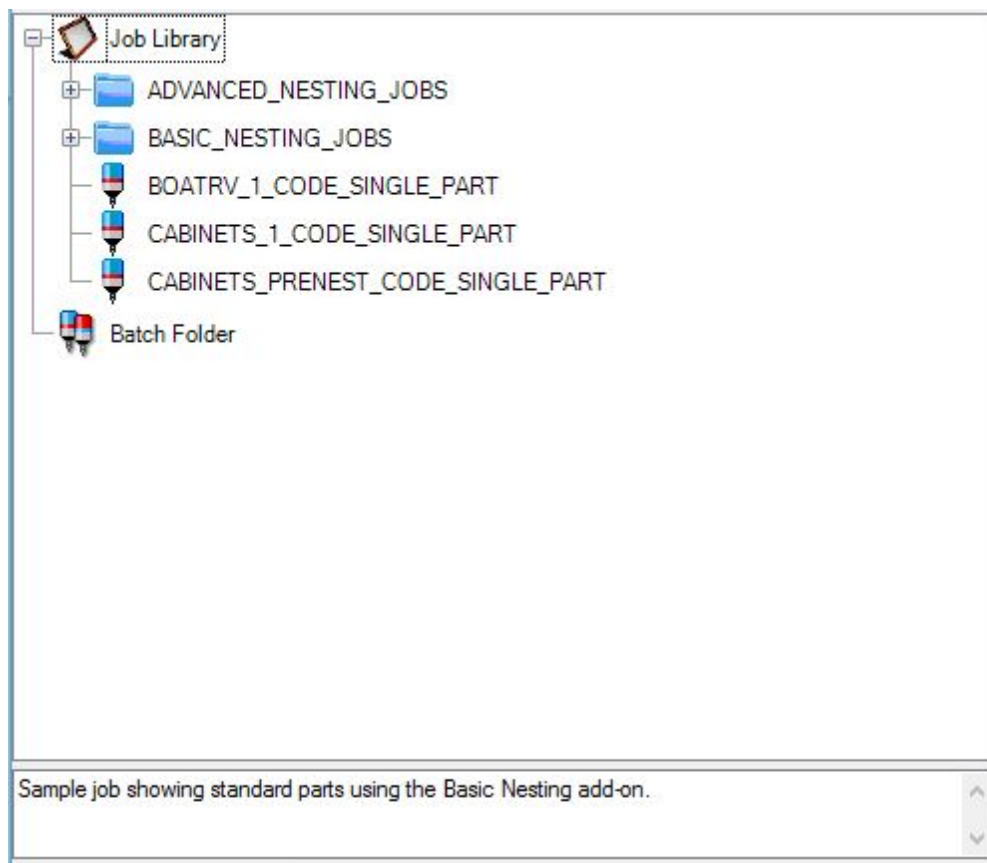
Run Job



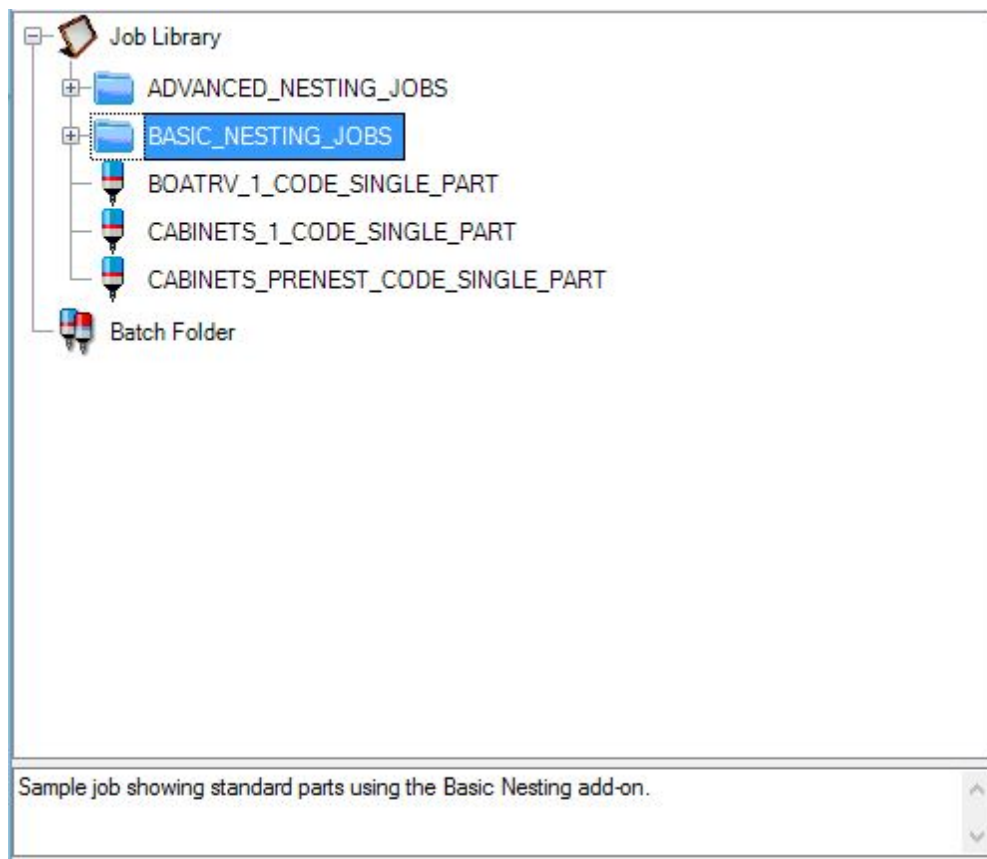
This button starts the selected job in Router-CIM Automation Suite and begins processing.

Folder Tree

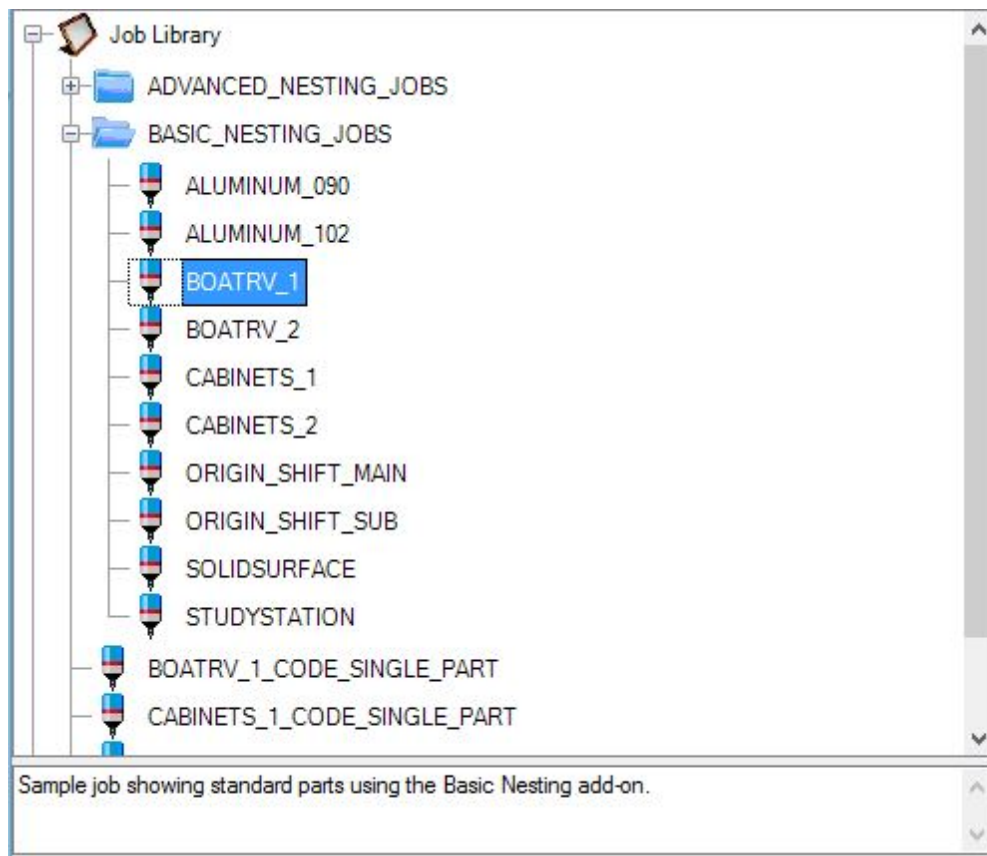
The folder tree is where your jobs are stored. Each job can be in the main folder or they can be stored in sub folders. The preference for how you store jobs is really up to you.



In this example, there is a folder called BASIC_NESTING_JOBS:



Selecting the BASIC_NESTING_JOBS folder opens the tree further to show the jobs that have been located there.



There could have been even more folders in this folder if the jobs needed to be separated further.

Batch Jobs

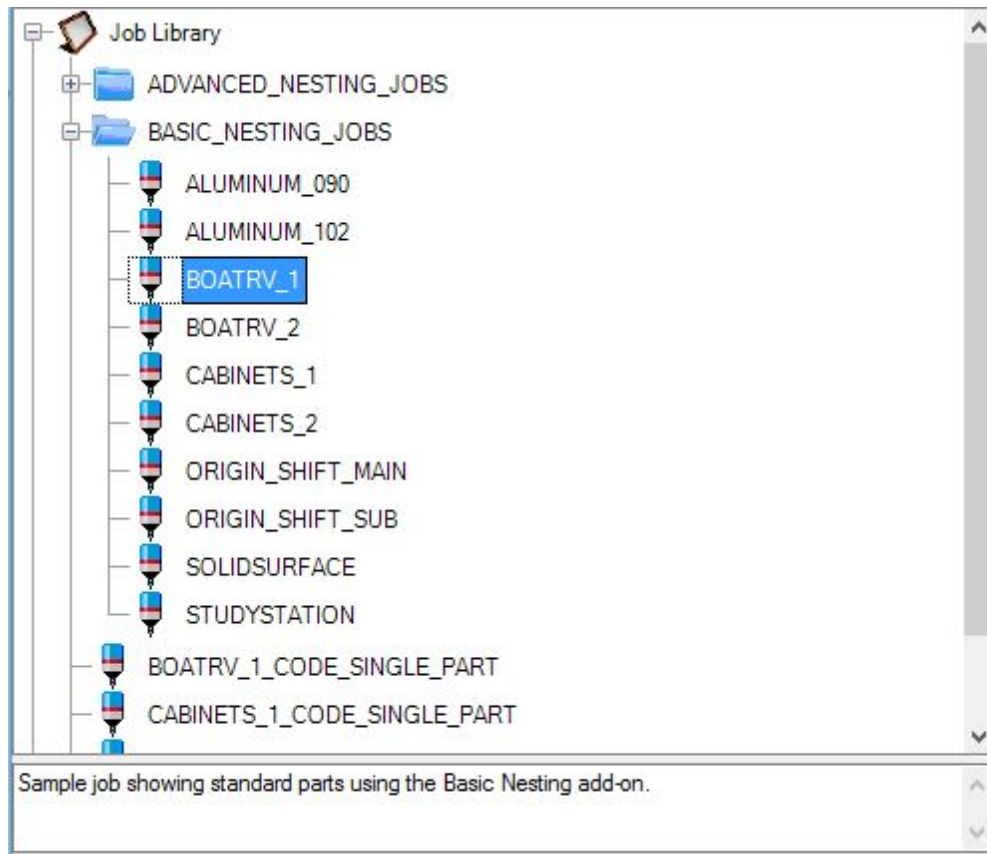
If you wish to load several jobs into Automation at one time, and have them executed in order, separately, then you can use Batch Processing. The process is relatively simple in that you would build your separate Router-CIM Automation Suite jobs as normal, and then add them (one at a time) into the batch folder. Once you create a batch job, you can run the entire batch and Router-CIM Automation Suite will run each job individually to completion, creating an output folder for each job, before going on to the next.

Batch Jobs are somewhat special and have a folder of their own. A batch job is really several jobs, each in one folder and run one at a time. Each job in a batch will run until it is done, and when the code is made for a particular job, then the next job will start up. None of the parts from one job are cut or nested with parts from another job.

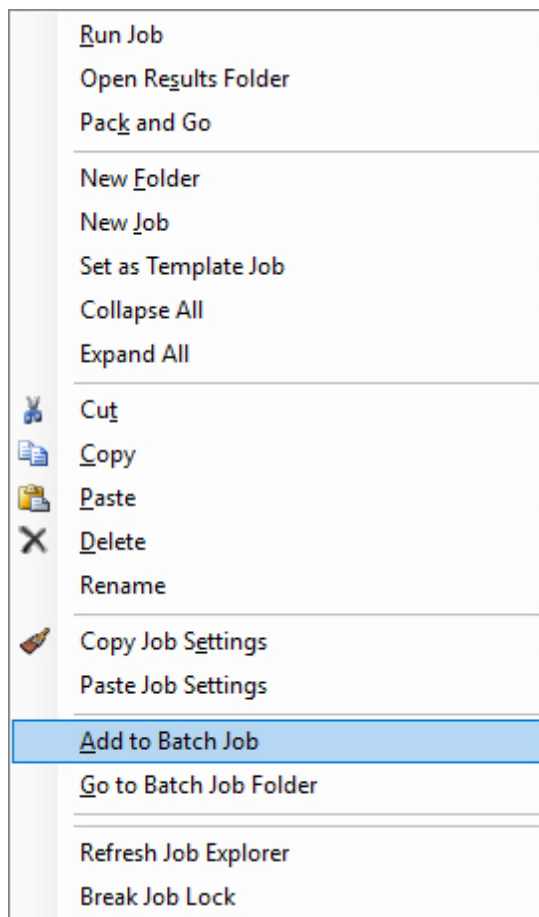
Note: If you close Router-CIM Automation Suite, the 'Batch Folder' will be cleared. If you want to keep the current batch, make sure to Save the batch.

To create a Batch Job, go to any folder containing jobs, and right click on a job and select '**Add to Batch Job**'.

For instance, using the BASIC_NESTING_JOBS folder created under the [Folder Tree](#) section. In this folder there are 10 jobs:

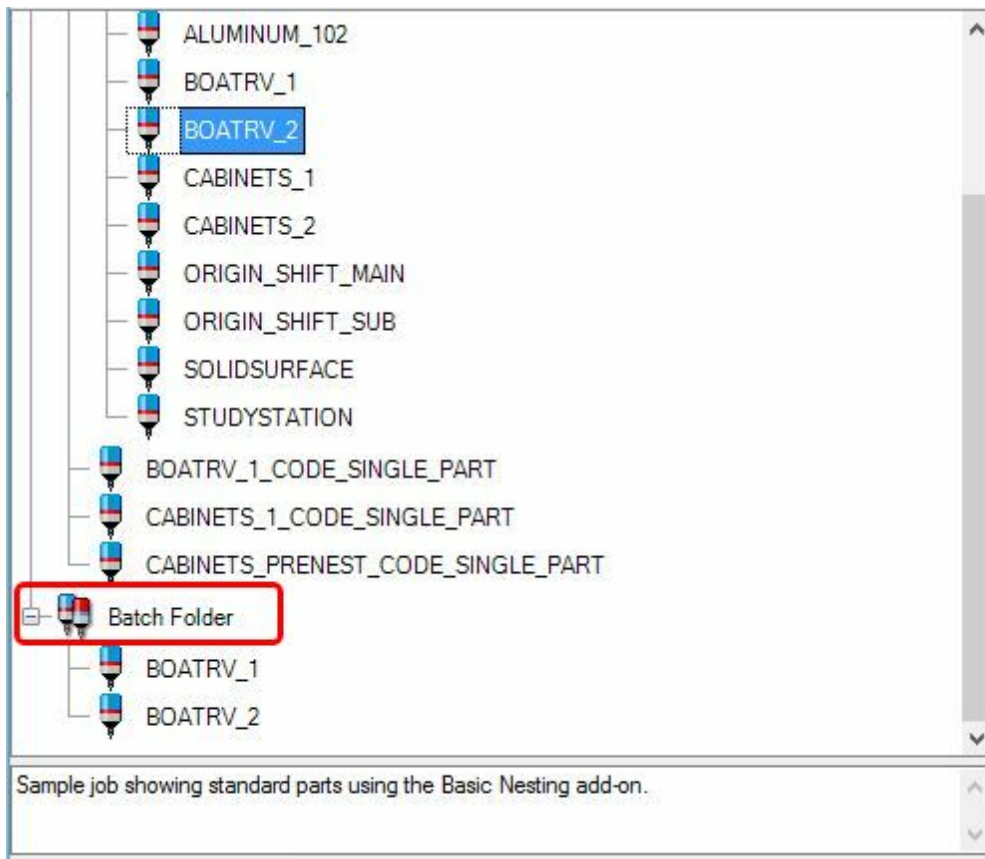


Select the BOATRV_1 job and Right-Click on it to bring up the menu, then select '**Add to Batch Job**'.

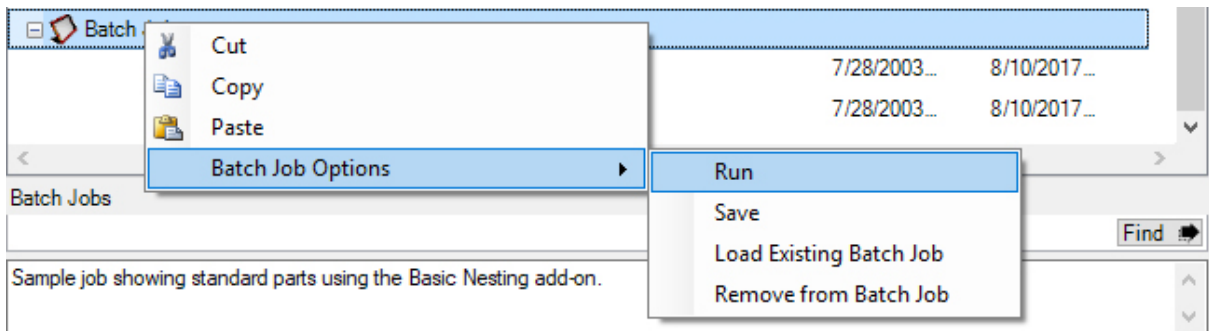


Now do the same for BOATRV_2 job in the folder.

Next, click back to the job library and select the 'Batch Folder' located at the bottom of the Folder Tree. You will see the 2 jobs listed under Batch Jobs when you select the + symbol next to the Batch Folder.



If you Right-Click on one of these jobs now you will see **'Batch Job Options'**.



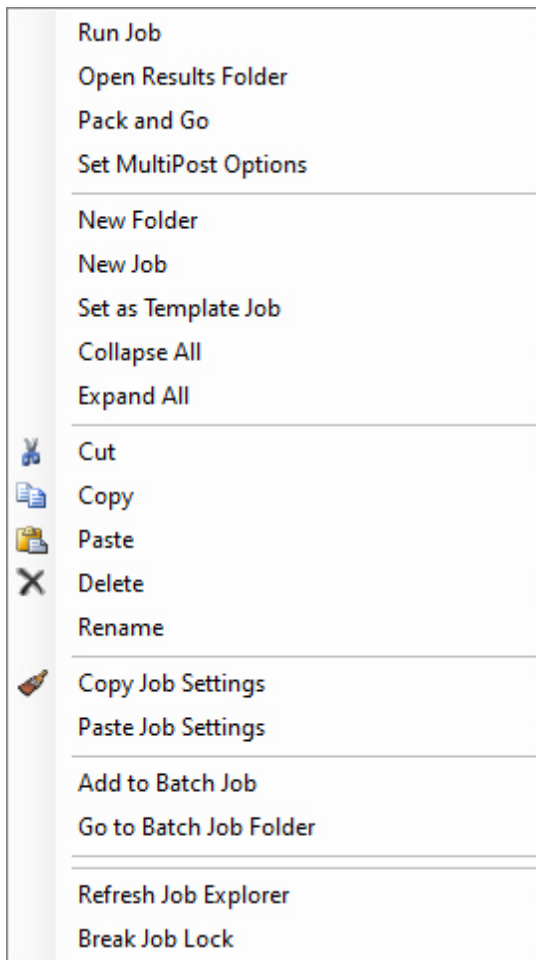
From here you can **'Run'** the batch, **'Save'** the batch to run it later, **'Load Existing Batch Job'** (if you have a previous batch job saved) or **'Remove from Batch Job'** to delete the job out of the Batch job folder.

Right Click Options for the Folder Tree

Router-CIM Automation Suite offers right click options for the jobs that are in the folder tree.

Note: For all the right click options, make sure to have the correct job selected before right clicking the job name.

Right Click Job Options



Run Job

This option starts the selected job in Router-CIM Automation Suite and begins processing.

Open Results Folder

Selecting this option will show you the result folder for a job, if a job is selected. If no job is selected, then the base results folder (the one containing all the others) is shown. The default results folder can be specified in the system settings under System Folders >> Data Output Folder.

Pack and Go

The Pack and Go function will allow you to package any job into just one file that can be sent to another user of Router-CIM Automation Suite who can then unpackage that job and have the same settings as you did on your Router-CIM Automation Suite system when the job was run.

For more information on Pack and Go, please refer to the [Pack and Go](#) section.

Set MultiPost Options

Please refer to the ['MultiPost Processing'](#) section.

New Folder

This option will create a new folder in the job tree.

New Job

This option will create a new job in the currently select folder.

Set as Template Job

This option will allow you to have a preset job setup for running a Right Click import. Multiple template jobs can be made but only one can be set to the default.

Collapse All

This option will close all the folders in the job tree.

Expand All

This option will open all the folders in the job tree.

Cut

Allows you to cut the folder or job.

Copy

Allows you to copy the folder or job.

Paste

Allows you to paste the folder or job. When you paste, select on the folder prior to right click and selecting paste.

Delete

This option deletes a job if a job is selected or an entire folder if a folder is selected. You cannot get these jobs or folders back once they are deleted, so be sure to have a pack and go of the job or a backup of the database if you think you might want them back at some point.

Rename

This option allows you to rename a job. This can also be accomplished by a slow double-click on the Job Name itself.

Copy Job Settings

This option allows you to copy all the Job Settings of the currently selected job. This includes the Post Processor, Knowledge Drawing and DOIT file as well.

Paste Job Settings

This option allows you to paste the copied Job Settings from another job and apply them to the currently selected job. This includes the Post Processor, Knowledge Drawing and DOIT file as well.

Add to Batch Job

This option allows you to create a copy of the currently selected job and it will add it to the Batch Folder.

For more information on the Batch Folder, click [here](#).

Go to Batch Job Folder

This option allows you to go to the Batch Folder.

For more information on the Batch Folder, click [here](#).

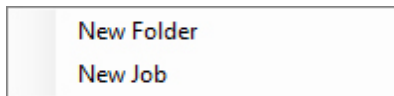
Refresh Job Explorer

This option will refresh the Job Tree. This is useful when using '[Shared Databases](#)'.

Break Job Lock

This option will allow you access to a job that is locked by another user when using '[Shared Databases](#)'.

Right Click Folder Tree Options



New Folder

This option will create a new folder in the folder tree.

New Job

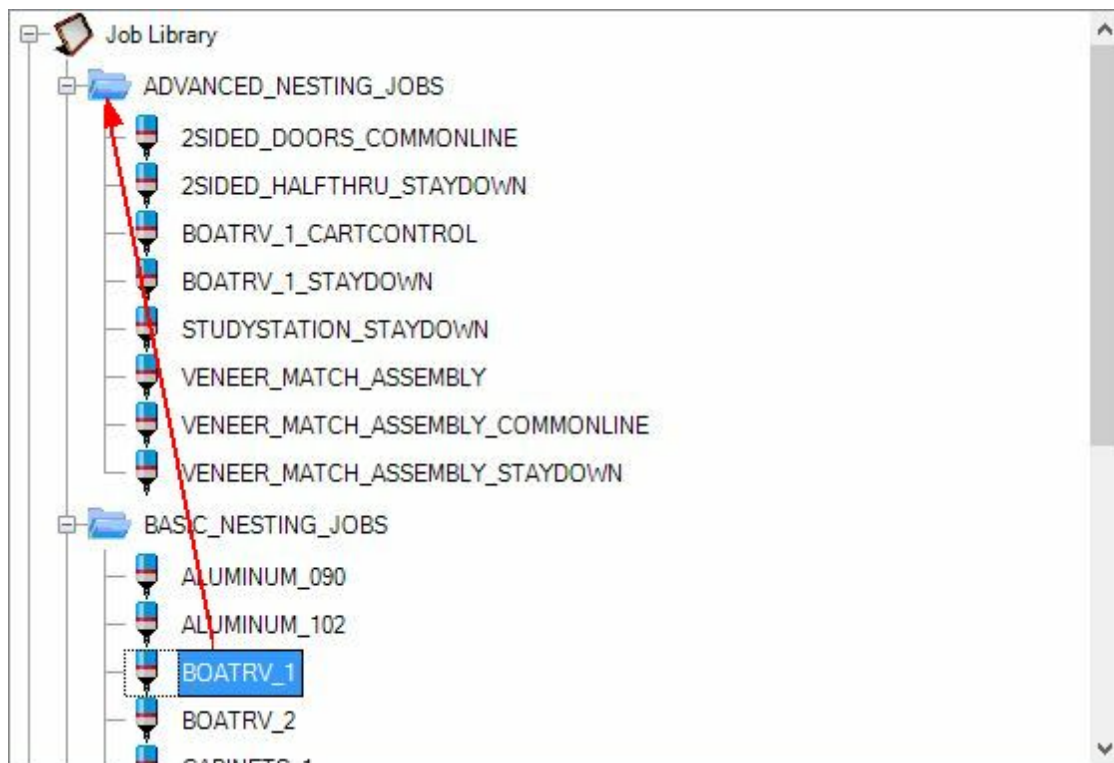
This option will create a new job in the currently selected folder.

Drag and Drop Job/Folder

Router-CIM Automation Suite supports the drag and drop feature for moving Jobs from folder to folder.

Simply select the job you want to move. Once the job is selected, left click on your mouse and hold. You will then be able to move the job to another folder.

For example if you wanted to move the sample job BOATRV_1 to the ADVANCED_NESTING_JOBS folder, select the BOATRV_1 job and than left click the BOATRV_1 job and hold the mouse button down.



Move to the ADVANCED_NESTING_JOBS folder and release the left mouse button.



You can see that BOATRV_1 is now in the ADVANCED_NESTING_JOBS folder.

This same procedure can be done to an entire folder.

Part Window

The Part Window contains all the settings for the parts inside the job. It is also the container for many of the job-specific settings. Each of these settings is available in one of the tabs at the top of the part window. These are available only when a job that contains parts is selected.

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
Part #	PartName	Qty	Material	Full path to part				
1	Sample1.dwg	4	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample1.dwg				
2	Sample2.dwg	1	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample2.dwg				
3	Sample3.dwg	2	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample3.dwg				
4	Sample4.dwg	2	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample4.dwg				
5	Sample5.dwg	4	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample5.dwg				
6	Sample6.dwg	1	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample6.dwg				
7	Sample7.dwg	3	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample7.dwg				
8	Sample8.dwg	3	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample8.dwg				
9	Sample9.dwg	3	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample9.dwg				
10	Sample10.dwg	3	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample10.dwg				
11	Sample11.dwg	3	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample11.dwg				
12	Sample12.dwg	3	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample12.dwg				

Variable Name	Effective Formula

The part window itself is editable for some of the part settings. You can change a parts Quantity, or Material directly from the part window by selecting the field and changing the value. Select a part and then select the quantity field. It will highlight indicating you can change the value. In this case we will change it from 4, to 8.

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
Part #	PartName	Qty	Material	Full path to part				
1	Sample1.dwg	4	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample1.dwg				
2	Sample2.dwg	1	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample2.dwg				
3	Sample3.dwg	2	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample3.dwg				
4	Sample4.dwg	2	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample4.dwg				
5	Sample5.dwg	4	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample5.dwg				
6	Sample6.dwg	1	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample6.dwg				
7	Sample7.dwg	2	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample7.dwg				

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
Part #	PartName	Qty	Material	Full path to part				
1	Sample1.dwg	8	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample1.dwg				
2	Sample2.dwg	1	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample2.dwg				
3	Sample3.dwg	2	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample3.dwg				
4	Sample4.dwg	2	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample4.dwg				
5	Sample5.dwg	4	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample5.dwg				
6	Sample6.dwg	1	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample6.dwg				
7	Sample7.dwg	2	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample7.dwg				

Changing the material is just as easy. You can pull down the materials available from the list on a selected part and change the material to another from the list.

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
Part #	PartName	Qty	Material	Full path to part				
1	Sample1.dwg	8	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample1.dwg				
2	Sample2.dwg	1	MatDesc	WGS\Sample2.dwg				
3	Sample3.dwg	2	HDPE_0.7500	WGS\Sample3.dwg				
4	Sample4.dwg	2	MAHOGANY_PLYWOOD_0.5000	WGS\Sample4.dwg				
5	Sample5.dwg	4	MAPLE_PLYWOOD_0.5000	WGS\Sample5.dwg				
6	Sample6.dwg	1	MATERIAL_0.2500	WGS\Sample6.dwg				
7	Sample7.dwg	3	MATERIAL_0.5000	WGS\Sample7.dwg				
8	Sample8.dwg	3	MATERIAL_0.7500	WGS\Sample8.dwg				
9	Sample9.dwg	3	MATERIAL_1.0000	WGS\Sample9.dwg				
10	Sample10.dwg	3	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample10.dwg				
11	Sample11.dwg	3	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample11.dwg				
12	Sample12.dwg	3	MATERIAL_0.5000	C:\rcim_work\Automation\Samples\DWGS\Sample12.dwg				

Selecting the a new material from the list changes the material that particular part is going to use.

Double-Clicking on any part will display the Part Properties. For more information on Part Properties, click [here](#).

Part Properties

Part Information

Part Name

C:\rcim_work\Automation\Samples\DWGS\Sample1.dwg

Record Desc

Material

MATERIAL_0.5000

Quantity

4

Maximize parts on sheet of material

Filler Qty

0

**This will change the quantity to -1

Label Information

Customer Info 1

ORD# 213423

Customer Info 2

Interior1

Customer Info 3

Customer Info 4

Customer Info 5

Customer Info 6

Customer Info 7

Customer Info 8

1

Inherit Label Data from Job

Part Options

Ignore Layer Panel

Start Point on longest side of part

☒

Manual Origin Part

Use ShapeDone Part when possible

Part Machining Operations

Nest and Code As Single Part

☒

Nest Part

☐

Code As Single Part

☐

Has an Associated Backside Macro

☐

Is an Irregular Stock Shape

☐

Output To Saw

☐

Macro Dimensions

Length

Width

Depth

Part Orientation

Rotate Part

☐

Rotate Angle

90

Mirror Part

☐

Mirror Axis

Horizontal

Nest Rotation

Same as Material

Save

Close

These are the part related properties that can be individually changed on any part in a job.

Part Properties

The Part Properties are broken down into two tabs. They are Part Information and Knowledge Information.

The screenshot shows the 'Part Properties' dialog box with the 'Part Information' tab selected. The 'Knowledge Information' tab is also visible. The 'Part Information' tab contains the following fields and options:

- Part Name:** C:\vcim_work\Automation\Samples\DWGS\Sample1.dwg
- Record Desc:** (empty field)
- Material:** MATERIAL_0.5000
- Quantity:** 4
- Filler Qty:** 0
- Maximize parts on sheet of material:** (unchecked)
- Label Information:**
 - Customer Info 1:** ORD# 213423
 - Customer Info 2:** Interior1
 - Customer Info 3:** (empty)
 - Customer Info 4:** (empty)
 - Customer Info 5:** (empty)
 - Customer Info 6:** (empty)
 - Customer Info 7:** (empty)
 - Customer Info 8:** 1
- Inherit Label Data from Job:** (unchecked)
- Part Options:**
 - Ignore Layer Panel:** (unchecked)
 - Start Point on longest side of part:** (checked)
 - Manual Origin Part:** (unchecked)
 - Use ShapeDone Part when possible:** (unchecked)
- Part Machining Operations:**
 - Nest and Code As Single Part:** (selected)
 - Nest Part:** (unchecked)
 - Code As Single Part:** (unchecked)
 - Has an Associated Backside Macro:** (unchecked)
 - Is an Irregular Stock Shape:** (unchecked)
 - Output To Saw:** (unchecked)
- Macro Dimensions:**
 - Length:** (empty)
 - Width:** (empty)
 - Depth:** (empty)
- Part Orientation:**
 - Rotate Part:** (unchecked)
 - Rotate Angle:** 90
 - Mirror Part:** (unchecked)
 - Mirror Axis:** Horizontal
 - Nest Rotation:** Same as Material

At the bottom right, there are 'Save' and 'Close' buttons, and a red 'X' icon.

Advanced Nesting Users:

If you have the Advanced Nesting Module, you will see an additional section as shown here for the Veneer Match Parameters:

The screenshot shows the 'Veneer Match Parameters' section with the following fields:

- Name:** (empty)
- Location Point:** (empty)
- Rotation:** (empty)

Go to the [Veneer Match Parameters](#) section for more information.

Part Information

Path Name

This box displays the path and name of the current selected part.

Record Description (Optional)

An editable field that will allow you to enter information you want to appear in the part description location of a label or a description you want to store in the job for the part if you do not use labels.

Material

Shows the material selected for the current part, and also allows you to change the part material from a drop-down list containing all the materials in your current database.

Quantity

Editable field to display or set the number of the selected part you want to appear in the job.

Maximize parts on sheet of material: If this checkbox is selected, when this part is nested, Router-CIM Automation Suite will change the quantity to -1. This will allow the nesting operation to maximize this part on a full sheet of material.

Filler Quantity (optional)

Allows you to set any number of this part to use to fill in gaps and open space in the nest. Use this option if you can store an extra quantity of the current part. Router-CIM Automation Suite will only generate filler parts if the option to allow them is turned on and there is sufficient room in the nest. No regular required parts from the job will be sacrificed to allow filler parts.

Label Information (optional)

There are 8 editable fields that can contain any data you think relevant to appear on the part labels. You can place data in as many or as few of these as you wish.

Inherit Label Data from Job

It is possible to have this data set to the same values for every part automatically if you set them in a job. This setting in each parts properties to 'Inherit Label Data from Job' will set all the label fields for each part to whatever they were set to in the job that the parts were added to.

Part Options

Ignore Layer Panel

When checked, this will ignore the default rectangle for the panel size on a macro, even if there is a layer to knowledge association present for it. It will further ignore that layer for nesting purposes.

Start Point on longest side of part

If checked, this option will move the start point of the shapes on the currently selected part to the mid point of whichever element on that particular shape is the longest, whether that is a line or an arc.

Manual Origin Part

This option is for Router-CIM macros only and will allow you to set where the origin of the part is instead of assuming 0,0.

Use ShapeDone Part when possible

Checking this option will allow the ShapeDone feature to be used on the selected part. Once the part is cut, that part and tool paths will be placed in the ShapeDone folder so that if it is cut at a later time, and has not been modified, it will simply be passed directly to nest and will not be cut again.

Nest and Code As Single Part

This option will allow all the parts in a job to have code created not only for the nests, but it will also generate one NC code file for each individual part in the job. See information below under

'Code as Single Part' if you want the individual NC code file to be offset by the materials edge allowance.

Nest Part

This will cause the parts in the job to be nested on the specified material.

Code As Single Part

Any part with this option checked will be cut and have code made for just the one part and it will not be considered for nesting. If you would like to have the part offset by the materials edge allowance like is done during the nesting process, please refer to the ['Code Single Material Edge Allowance'](#) section. In order to use this feature, the outside geometry of the part must be on a layer defined under the Layer Lists in your ['General Settings'](#).

Has an Associated Backside Macro

This option is for Router-CIM macros only and will cause Router-CIM Automation Suite to look for a macro with the same name, followed by -b. The backside macro will then be cut as a single part (not nested) and the code included in the results folder. For instance Style1.scn has a backside macro named Style1-b.scn.

If Style1.scn were a part in a job, you could check this option and both Style1.scn and Style1-b.scn would be cut, but only Style1.scn would be nested. The code for Style1-b.scn would be as a single part.

Is an Irregular Stock Shape

A drawing that has geometry on layer IR_STOCK can be placed in a job and then if this option is checked, that shape would be used as a piece of material for nesting. Any shape is allowed, but it should be a closed polyline (on layer IR_STOCK).

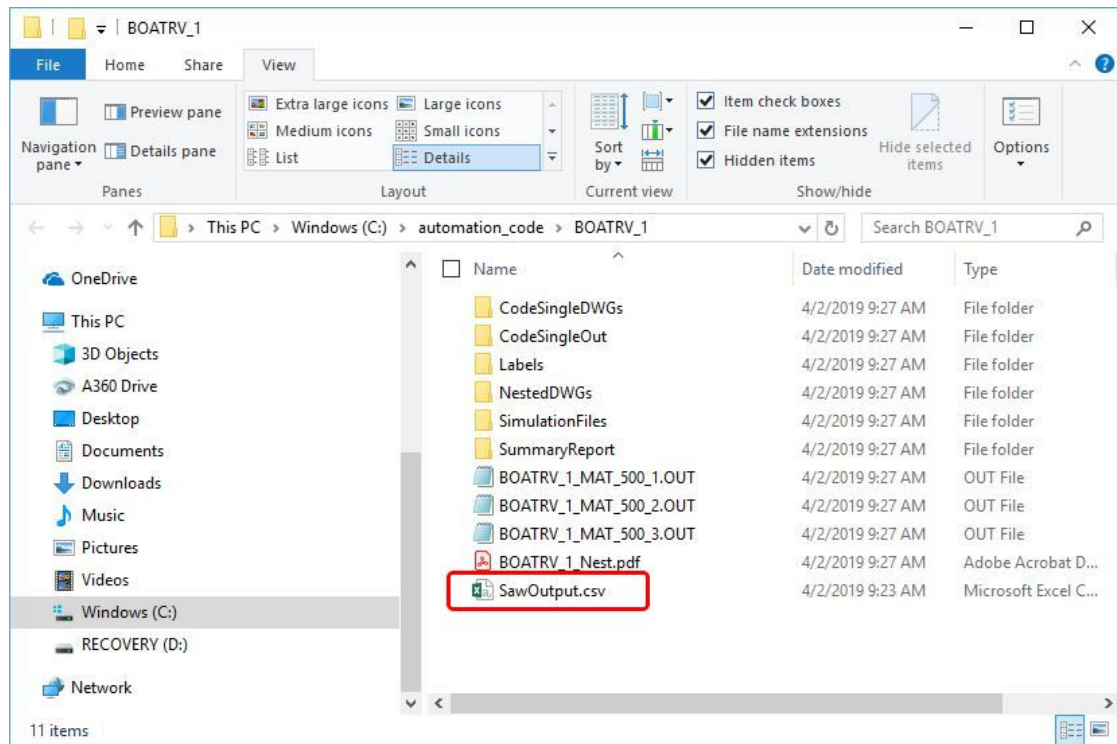
Output to Saw

This option will output a CSV that will include the information needed to feed to a saw optimization software.

The information included in the CSV is as follows:

Column A: Part Name (Part Location)
Column B: Material Description
Column C: Quantity
Column D: Record Description
Column E: Material Code
Column F: XDim
Column G: YDim
Column H: ZDim
Column I: Open
Column J: Open
Column K: Open
Column L: Open
Column M: Part Name without the extension
Column N: Router-CIM Part Number

The CSV file created will be placed in the 'Output' folder from Router-CIM Automation Suite.



Macro Dimensions

Generic Record X Dim

When a Router-CIM macro is used, the X dimension setting can be overridden with this field.

Generic Record Y Dim

When a Router-CIM macro is used, the Y dimension setting can be overridden with this field.

Generic Record Z Dim

When a Router-CIM macro is used, the Z dimension setting can be overridden with this field. Note that the material can be used for the Z value to determine a parts thickness.

Part Orientation

Rotate Part

Checking this option will force the part to rotate to the angle set in the Rotate Angle field (to the right). Whether or not the material has part rotation set, this field will override the setting and rotate the part from the default view it was drawn in. This setting has no effect on the material rotation angle and any material related settings will occur during the nesting section of a job run.

Only Pattern Recognition will stop this rotation. If a part is being drilled with Pattern Recognition and has more than 1 hole drilled at a time (in other words a pattern of two or more) then the part will be rotation locked and cannot be rotated on the material.

Rotate Angle

Controls the rotation angle of the part when an override to the rotation angle is needed. See above. Based on AutoCAD angle definitions.

Mirror Part

Checking this option will force the part to be mirrored about the axis specified to the right (Mirror Axis).

The mirroring is done prior to cut and it is entirely possible to have a part mirrored and rotated using the rotate commands within this section.

Pattern Recognition has no effect on mirror since the part will still have the orientation of its holes in the same direction.

Mirror Axis

Controls the direction of the mirror. Available options are Vertical or Horizontal. The Mirror Part box must be checked to turn this option on.

Nest Rotation

This is the default setting for the current part nested on a sheet. You can allow it to rotate based on the material settings or force it to override the material settings and control the rotation angle. The available options are **Same as Material, ALL, 0, 0 90, 0 180, 0 90 180**. Once this setting is changed, the part rotation will be set to this setting and ignore the material setting defined in the Material Database.

NOTE: It is possible to have this rotation set as well as the Rotate Part box checked and multiple angle rotations will occur.

To scale your parts from inch to metric or metric to inch prior to processing the job, please go to ['Scale Part Geometry in Router-CIM Automation Suite'](#).

Knowledge Information

It is possible to have certain parts in a job nested and cut on one machine, and single parts (not nested) cut on another machine. These single parts can be secondary operations to a particular part, or just some parts that you wish to cut one at a time on a different machine. These parts cannot be nested. They can use a different knowledge drawing, DOIT file, Post Processor, and different part settings. They all, however, must be set to **'Code as Single Part'**.

Part Properties

Part Information | **Knowledge Information**

Primary (Front Side) Knowledge Configuration

Knowledge
WOOD_SCIM_RS.dwg ☒ Inherited

DOIT File
SCIM_WOOD_DOIT.dat ☒ Inherited

Machine Post
RS274D.\$PP ☒ Inherited

Restore Inherited Values

Secondary (Back Side) Knowledge Configuration

Knowledge
WOOD_SCIM_RS.dwg ☒ Inherited

DOIT File
SCIM_WOOD_DOIT.dat ☒ Inherited

Machine Post
RS274D.\$PP ☒ Inherited

Restore Inherited Values

Save Close

Primary Knowledge Configuration

When a job has parts that are marked as Code as Single Part, the Knowledge Information screen will be available. You can set the Primary Knowledge in this window to run with a different knowledge drawing, DOIT file, and post processor than the other parts that are being cut in the main job.

Secondary Knowledge Configuration

This is only for Router-CIM macros, where the part properties are marked "Has an Associated Backside Macro". The backside macro (same name as the regular macro, with a -b at the end) can be cut with a different knowledge drawing, DOIT file, and post processor than the front side (nested side) of the same macro.

When using this feature, you should uncheck the box marked "Use Primary Knowledge for Secondary Operations" if you need a secondary knowledge drawing, and also the "Use Primary Machine for Secondary Operations" to specify the secondary post processor.

Knowledge and Post Info Preview

Primary Knowledge: WOOD_SCIM_RS.dwg [New] [Edit]

Secondary Knowledge: WOOD_SCIM_RS.dwg [Edit]

☐ Use Primary Knowledge for Secondary Operations

Primary Machine: RS274D.\$PP [New] [Edit]

Secondary Machine: RS274D.\$PP

☒ Use Primary Machine for Secondary Operations

Material: MATERIAL_0.5000 [New] [Edit]

DOIT File: SCIM_WOOD_DOIT.dat [New] [Edit]

Inherited

Checking the Inherited box in any of these fields will set that particular property to the same settings as the main window is using for the job.

Job Settings

The Job Settings tab in a job contains the variables that affect various aspects of the job during the automation run. There are several sections to these settings.

Parts Job Settings Nesting Advanced Nesting Dynamic Variables Printing and Labels Part Name Output Files Results

General Options

☐ Output DXF for laser ☐ Part Multiplier 1

☐ Enable ShapeDone

☐ Accelerate/Decelerate ☐ Code Single Material Edge Allowance

☒ Flatten Splines

Sequence Sorting Options: RANK, TOOL SORT, CLOSEST POINT

Geometry Scale Factor: 1.0000000000 Start Point Override: SP_

Small Part Options

☒ Rename Outside Layers

☐ Enable Cutter Bridge

Area Tolerance (SML): 100.0000

Dimension Tolerance (SML): 3.0000

Area Tolerance (SMX): 0.0000

Dimension Tolerance (SMX): 0.0000

New Part Defaults

Part Nesting Options

☒ Nest and Code as a Single Part

☐ Nest Part

☐ Code as a Single Part

☒ Start Point on Longest Side of feature

☒ Use ShapeDone

Default Quantity: 1

Default Filler Quantity: 0

Filler Parts

☐ Fill Entire Sheet

☒ Fill Enclosing Rectangle

Small Inside Feature Options

☒ Rename Inside Layers

Area Tolerance (ISML): 10.0000

Dimension Tolerance (ISML): 1.0000

Area Tolerance (ISMX): 2.0000

Dimension Tolerance (ISMX): 0.5000

General Options

Output DXF for laser

This option will create a DXF file of each nest or single part cut for a projection laser to use in order to show the layout of the parts on the table. If you do not have a projection laser, leave this option off.

Enable Shape Done

Enabling Shape Done will turn on Shape Done for the job. If parts have the option turned on to enable Shape Done, then a copy of the part will be stored with the knowledge drawing.

Accelerate/Decelerate

Enabling Accelerate/Decelerate will use any cutting knowledge that has the ACC/DEC option selected for machines that need the NC code to slow down and speed up the machine.

Flatten Splines

Splines are special objects in that they have a continuously changing radius. Being such, they must be converted into arc and/or line segments to be able to produce NC code. If the AutoCAD Express Tools are installed, the FLATTEN command is available and Router-CIM will use it to convert the spline. You can also use the command manually in AutoCAD by typing FLATTEN at the command line or through the Express Tools menu or ribbon. If the FLATTEN command is not available, Router-CIM will convert the spline into small line segments for NC processing. If the FLATTEN command is failing to convert the spline, you may want to disable it from being used in Router-CIM Automation Suite.

Enabling Flatten Splines allows Router-CIM to convert the spline feature into arc and/or line segments.

Sequence Sorting Options

You may select from any of the sorting options listed and that option will be used to sort all the parts in the job during Sequence.

Geometry Scale Factor

The value entered into this field will be used during the Router-CIM Automation Suite job run when the parts is loaded into AutoCAD. For instance, to scale a part drawn in inches to mm, you could use a value of 25.4.

Note: This field will only affect the part drawing itself, cut knowledges and materials will need to be configured for the resulting part size after the scale factor has been applied.

Part Multiplier

The Part Multiplier will multiply the part quantities by the selected value when a job is processed. To activate, check the box and use the up and down arrows to set the part multiplier.



☐ Part Multiplier 1



☒ Part Multiplier 4

Code Single Material Edge Allowance

Enabling Code Single Material Edge Allowance allows for a part that is set to 'Code as Single Part' to have the individual NC code file produced with the Material Edge Allowance that is set up in your Materials Database.

Start Point Override

Start Point Override will change the startpoints on geometry in Router-CIM Automation Suite during a job run, you can set it to SP_ (or any set of characters up to a 50 character limit) including the use of the * (asterisk) wildcard. The * (asterisk) is a way to ignore/discard any information that comes after it. An example would be if you were trying to match to multiple layers such as SML and SMX parts, you could use SM* in this field. When Router-CIM Automation Suite runs the job and sees a layer with SM at the beginning of the layer name, the rest of the layer name will be ignored and it will use the Start Point Override option. In the case where the user has defined the layer prefix then rename your layers with that prefix and put the prefix used in this field. Once that has been completed, you will need to change the DOIT file layer to knowledge associations in order to cut the layers properly. You can then run the job, Router-CIM Automation Suite will stop on the geometry on those layers and prompt you to pick a start point.

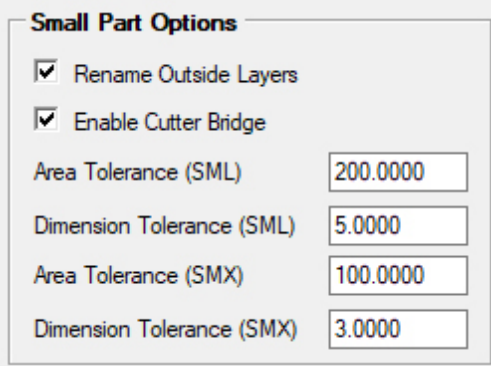
Small Part Options

Rename Outside Layers

If this option is active, parts will have the outside layer rename with a prefix of SML or SMX so that an alternate knowledge can be used on them. This is useful if you cut small parts and they tend to lose vacuum or move.

Rename Outside Layers uses two variables. The first is 'Area Tolerance' which is based on the square units of the outside geometry. If the outside geometry is equal to or less then the value, it will be considered an SML or SMX part and the outside layer name will receive the prefix SML or SMX. The second variable checks for the long, narrow parts. A line is drawn at the 'Dimension Tolerance' in X and Y when the part is processed. If the part, as it is oriented, does not intersect the line, it will also be considered an SML or SMX part even if the area of the outside geometry is over the Area Tolerance.

The options allows for 2 sets of small parts. SML parts will be defined by 'Area Tolerance (SML)' and 'Dimension Tolerance (SML)'. SMX parts will be defined by 'Area Tolerance (SMX)' and 'Dimension Tolerance (SMX)'



Small Part Options	
<input checked="" type="checkbox"/> Rename Outside Layers	
<input checked="" type="checkbox"/> Enable Cutter Bridge	
Area Tolerance (SML)	200.0000
Dimension Tolerance (SML)	5.0000
Area Tolerance (SMX)	100.0000
Dimension Tolerance (SMX)	3.0000

For example, if a part is less then the area for the 'Area Tolerance (SML)' but an area greater than the 'Area Tolerance (SMX)' you have indicated and this part has an outside layer such as OUTSIDE_0.7500, this feature will rename that layer to SMLOUTSIDE_0.7500. This way in the

DOIT file you can make knowledge associations to the layer SMLOUTSIDE_0.7500 so that the small parts are machined differently then larger parts. In this same example, if the part has an area of 250 but it is a 4x75 rectangle, it will also rename the outside layer to SMLOUTSIDE_0.7500 because of the dimension tolerance.

Also then if a part is less then the area for the 'Area Tolerance (SMX)' you have indicated and this part has an outside layer such as OUTSIDE_0.7500, this feature will rename that layer to SMXOUTSIDE_0.7500. This way in the DOIT file you can make knowledge associations to the layer SMXOUTSIDE_0.7500 so that the very small parts are machined differently then large or small parts. In this same example, if the part has an area of 150 but it is a 2x75 rectangle, it will also rename the outside layer to SMXOUTSIDE_0.7500 because of the dimension tolerance.

Note: Any layer named exactly PROFILE_OUTSIDEDOOR and PROFILE_OUTSIDEDOORSMALL will not use the SML/SMX feature.

Enable Cutter Bridge

If this option is active any cutting knowledge that has a cutter bridge identified and is used, the cutter bridge amount will be added to the material's bridge width in order to give more material around the specified part.

For more information on setting a cutter bridge in a cut knowledge, refer to the ['Cutter Bridge'](#) section under ['Tool Information'](#).

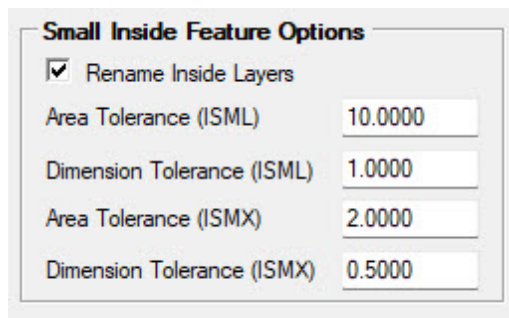
Small Inside Feature Options

Rename Inside Layers

If this option is active, parts will have the inside layer rename with a prefix of ISML or ISMX so that an alternate knowledge can be used on them. This is useful if you cut small inside features to automate the process of identifying the small inside features.

Rename Inside Layers uses two variables. The first is 'Area Tolerance' which is based on the square units of the geometry. If the geometry is equal to or less then the value, it will be considered an ISML or ISMX part and the layer name will receive the prefix ISML or ISMX. The second variable checks for the long, narrow features. A line is drawn at the 'Dimension Tolerance' in X and Y when the part is processed. If the feature, as it is oriented, does not intersect the line, it will also be considered an ISML or ISMX part even if the area of the geometry is over the Area Tolerance.

The options allows for 2 sets of small inside features. ISML parts will be defined by 'Area Tolerance (ISML)' and 'Dimension Tolerance (ISML)'. ISMX parts will be defined by 'Area Tolerance (ISMX)' and 'Dimension Tolerance (ISMX)'



Small Inside Feature Options	
<input checked="" type="checkbox"/> Rename Inside Layers	
Area Tolerance (ISML)	10.0000
Dimension Tolerance (ISML)	1.0000
Area Tolerance (ISMX)	2.0000
Dimension Tolerance (ISMX)	0.5000

For example, if an inside features is less then the area for the 'Area Tolerance (ISML)' but an area greater than the 'Area Tolerance (ISMX)' you have indicated and this part has an inside feature layer such as CUTOUT_0.7500, this feature will rename that layer to ISMLCUTOUT_0.7500. This way in the DOIT file you can make knowledge associations to the layer ISMLCUTOUT_0.7500 so that the small inside features are machined differently then larger inside features. In this same example, if the inside feature has an area of 25 but it is a 0.75x75 rectangle, it will also rename the outside layer to ISMLCUTOUT_0.7500 because of the dimension tolerance.

Also then if an inside feature is less then the area for the 'Area Tolerance (ISMX)' you have indicated and this inside feature has a layer such as CUTOUT_0.7500, this feature will rename that layer to ISMXCUTOUT_0.7500. This way in the DOIT file you can make knowledge associations to the layer ISMXCUTOUT_0.7500 so that the very small inside features are machined differently then large or small inside features. In this same example, if the inside feature has an area of 15 but it is a .375x75 rectangle, it will also rename the outside layer to ISMXCUTOUT_0.7500 because of the dimension tolerance.

New Part Defaults

Part Nesting Options

Nest and Code as Single Part

Checking this option will cause Router-CIM Automation Suite to nest each part in the job, and also make a separate NC program for each and every part on its own. This is useful for situations where a part is later broken or lost, a piece of material can be placed on the machine at the home position and run just the program for the one part.

Nest Part

When enabled, this option will cause Router-CIM Automation Suite to place every part in the job into a nested sheet based on the material specified. If more than one material exists for parts in a job, nests will be created for each sheet with the parts specified for that material.

Code as Single Part

Using this setting will cause Router-CIM Automation Suite to make a separate NC program for each part in a job. No parts will be nested together.

NOTE: For more information about 'Code as Single Part', please visit the ['Part Properties'](#) section.

Start Point on Longest Side of Feature

Using this option will ensure that when new parts are added to a job, the Start Point on Longest Side of Feature box is checked by default.

Checking this box will force Router-CIM Automation Suite to move the startpoint to the longest side of the feature, regardless of where the startpoint exists currently unless using [Start Point Control](#).

Use ShapeDone

This option must be turned on in order for a part to be passed to the ShapeDone folder and stored.

NOTE: ShapeDone will become inactive when parts are set to 'Nest and Code as Single Part' or 'Code as Single Part'. For more information on ShapeDone, click [here](#).

Default Quantity

This field allows you to set the default quantity that parts are set to when they are added to a new job. When a new job is created, the number of parts that are set automatically will be set to this number.

Default Filler Quantity

This parameter is to set the default number of filler parts set for each new part in a job. If there is enough space in the nest, then these parts will be placed in the nest to fill the space and boost sheet yield, up to the number set in this section.

Filler Parts Control

Allows you to set any number of part(s) to use to fill in gaps and open space in the nest.

When a filler part quantity is defined, you can fill the nest based on the sheet size or the enclosing rectangle which is the amount of area used by the nested parts themselves.

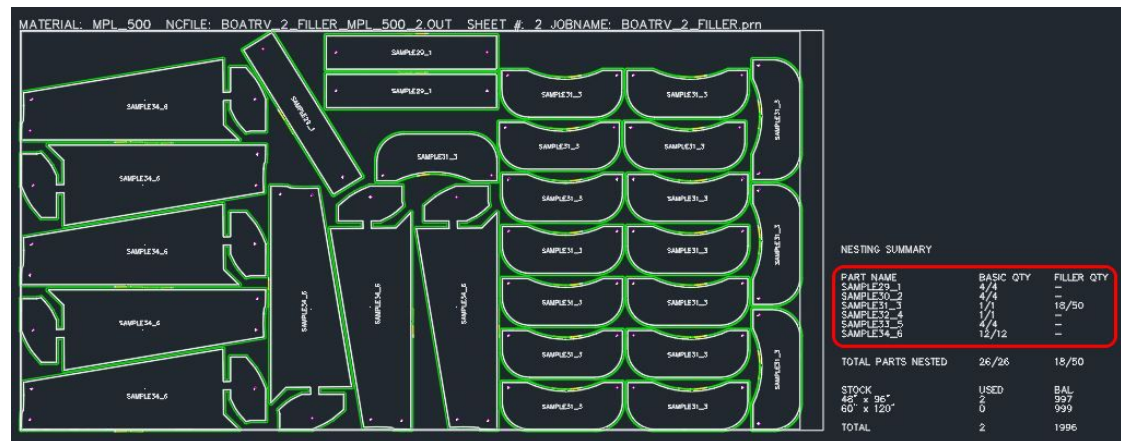
Fill Entire Sheet:

Filler Parts

☒ Fill Entire Sheet

☐ Fill Enclosing Rectangle

When set to 'Fill Entire Sheet', if filler parts are defined, the scrap settings for the material may be ignored and nesting will fill any left over area on a sheet with filler parts if the quantity allows.



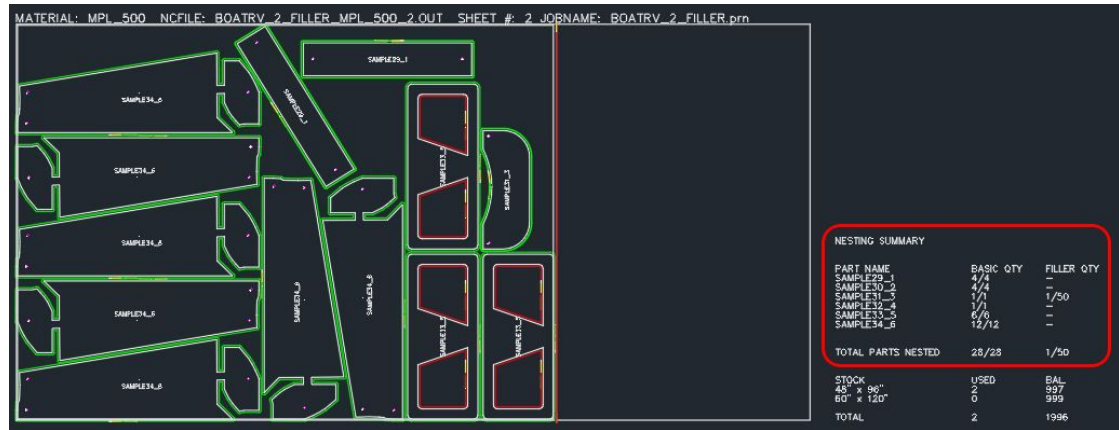
Fill Enclosing Rectangle:

Filler Parts

☐ Fill Entire Sheet

☒ Fill Enclosing Rectangle

When set to 'Fill Enclosing Rectangle', if filler parts are defined, the scrap settings for the material will not be ignored and nesting will fill any left over area within the enclosing rectangle created by the nested parts.



For more information on how to set the Filler Parts quantity, refer to the ['Part Properties'](#) section.

Nesting

Note: In order for nesting to know the boundary of the parts, you will need to make sure that you have defined the Outside layer under the ['Layer Lists'](#) tab in the ['Settings'](#) for Router-CIM Automation Suite.

Parts Job Settings Nesting Advanced Nesting Dynamic Variables Printing and Labels Part Name Output Files Results

Nested Layout Presentation

Labels

☒ Label Parts

☒ Label Repeated Parts

Text Size

Display Point

☐ Center of Gravity ☒ Insertion Point

Text Display

☐ Horizontal Only

☒ Longest Edge of Part

☒ Follow Part Rotation

☒ Reduce Scale to Fit

Tolerance Angle (Degrees)

Text Length Scale Factor

Text Width Scale Factor

Auto Text Height Scale

Summary Report

☒ Include Summary Report

☐ Show Final Summary

Text Size

Length

Location

List Stocks

Nested Layout Spacing

Horizontal Spacing

Vertical Spacing

Layouts per Row

☒ Display Repeated Layouts

Accuracy

Measurement

Packing Direction Control

☒ Auto

☐ Horizontal Packing

☐ Vertical Packing

Nesting Start Point

☐ Top Left

☒ Top Right

☐ Bottom Left

☐ Bottom Right

Sheet Origin

☐ Top Left

☒ Top Right

☐ Bottom Left

☐ Bottom Right

☒ Save Nests or Parts as Drawings

☒ Save AutoNEST Parts and Tasks

Part Inside Part

☒ Enable Part Inside Part Nesting

Layer

No Cut Layer

Area

X Minimum

Y Minimum

Nesting Engine:

Nested Layout Presentation

This section contains parameters that affect the look of the nested sheet, summaries, and part labels on the nest.

Labels

Label Parts

If you want the parts in the nest labeled, check this option.
This option is ON by default.

Label Repeated Parts

Check this option when you want each part in the nest labeled, even when it is duplicated in the nest pattern.
This option is ON by default.

Text Size

Set the text size (in current units) for the text that appears on the part labels.
Default value is 0.75"

Display Point

Center of Gravity: Place the part label at the center of gravity of the part.

Insertion Point: Place the part label at the insertion point of the part.

Using NC Variables to Control Label Placement: 3 variables need to be added to the [NCVar Editor](#). The Nestpoint variable is designed to try and find a location on the part that does not intersect any other geometry of the part.

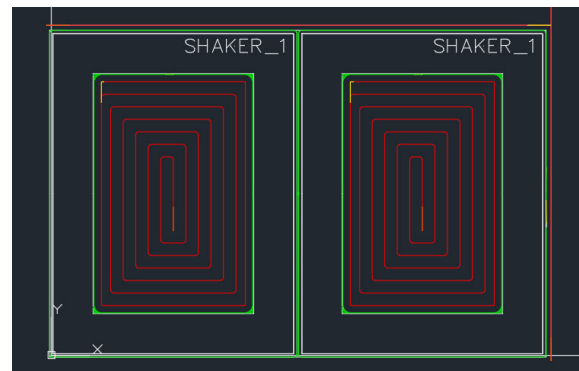
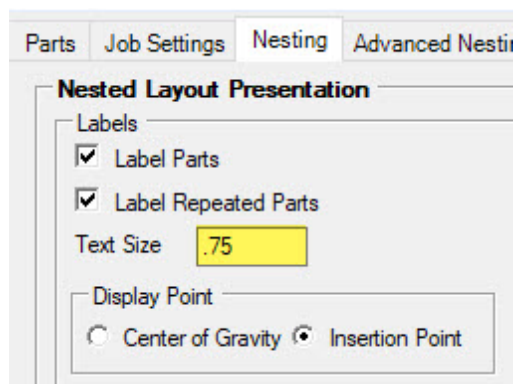
Variable Example Value:

Name:

NESTPOINT T or Nil (Required)
label_size_x 5.0 (Numeric value, Optional)
label_size_y 2.0 (Numeric value, Optional)

When using the *NESTPOINT* variable only, the 'Text Size' field located under 'Label Parts' will drive the size of the rectangle that will be searched for on the part to fit the part label value.

In this example, an area of 0.75 x 0.75 will be defined in order to place the part label with the resulting location on the part:



You can also use the *label_size_x* and *label_size_y* variables if you want to define your own rectangle size without affecting the size of the text. See above for adding these two variables to the NCVars of Router-CIM Automation Suite.

Note: Insertion Point needs to be the selected option for Display Point when using the *NESTPOINT* variable.

Text Display

Horizontal Only: This will place the label unscaled and horizontal only based on the center of gravity of the part only.

Note: The 3 check boxes and the 4 text boxes will be grayed out if this option is selected

Longest Edge of Part: Aligns the label with the longest edge of the part.

Follow Part Rotation: Keeps the label aligned with the part even if the nesting function rotates the part.

Reduce Scale to Fit: Reduces the scale of the text to fit within the part.

Tolerance Angle (degrees): Degrees in which the text will snap to the nearest X or Y axis.

Text Length Scale Factor: Text Height is computed from the center of gravity of the Part to the nearest edge of the Part's (smallest) enclosing rectangle. The computed Text Height is reduced by the scale factor (0.8 default). This scale factor can be adjusted depending on the FONT used in the drawing to achieve a visually pleasing result.

Text Width Scale Factor: Text Height is computed from the center of gravity of the Part to the furthest edge of the Part's (smallest) enclosing rectangle. The computed Text Height is reduced by the scale factor (0.95 default). This scale factor can be adjusted depending on the FONT used in the drawing to achieve a visually pleasing result.

Auto Text Height Scale: When the specified Text Height is "Auto", each text height is calculated based on the smaller width of the Part's enclosing rectangle multiply by this factor, 0.15 default.

Summary Report

Include Summary Report

The Summary Report is a recap for each nest including utilization, enclosing rectangle and parts nested on the sheet.

NESTING REPORT OF
TASK NAME : MAT_500

STOCK	STOCK #1
QTY	1
COST	0.00
STOCK SIZE	48" x 96"
ENCLOSING RECT	48.2" x 96.1"
ENCLOSING RECT/ STOCK AREA	4637.1/ 4608 100.63%
TOTAL PART AREA/ ENCLOSING RECT	2954.5/ 4637.1 63.71%
UTILIZATION	64.12%
TOTAL PART PERIMETER	1203.9"
PART NAME	
2	1/1 0/1
4	2/2 0/2
7	2/3 1/3
8	1/3 2/3
9	2/3 1/3
10	3/3 0/3
11	3/3 0/3
12	1/3 2/3
SUB TOTAL	15/32

Show Final Summary

The Final Summary shows all the parts nested for the material and the number of sheets used. It is located with the last nest.

NESTING SUMMARY

PART NAME	BASIC QTY	FILLER QTY
1	4/4	-
2	1/1	-
3	2/2	-
4	2/2	-
5	4/4	-
6	1/1	-
7	3/3	-
8	3/3	-
9	3/3	-
10	3/3	-
11	3/3	-
12	3/3	-
TOTAL PARTS NESTED	32/32	-
STOCK	USED	BAL.
48" x 96"	3	9996
48" x 120"	0	999
60" x 96"	0	999
60" x 120"	0	999
TOTAL	3	12993

Text Size

This setting controls the text size of the summary report.

Location

The location of the Summary Report can be set to either RIGHT, TOP-LEFT, or TOP-RIGHT.

Length

This is the number of characters allowed before proceeding to another line.

Accuracy

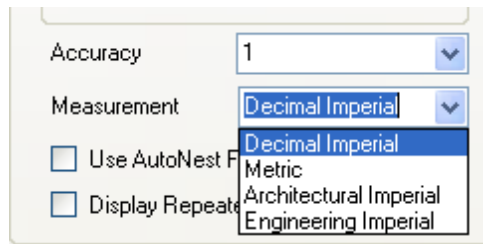
This is a setting for the decimal places that will show for the numbers in the nest summaries. For instance using 1 will show a material as 96.0x60.0. Using more than one will increase the number of decimal locations that are reported. 1-4 are the only possible inputs in the list.

The screenshot shows a settings window with the following elements:

- Accuracy:** A dropdown menu currently showing '1'. The menu is open, displaying options 1, 2, 3, and 4.
- Measurement:** A dropdown menu located directly below the Accuracy dropdown.
- Use AutoNest:** A checkbox that is currently unchecked.
- Display Repeated Layouts:** A checkbox that is currently unchecked.

Measurement

The measurement type will be used by the font that displays the units used in the nesting and summaries.



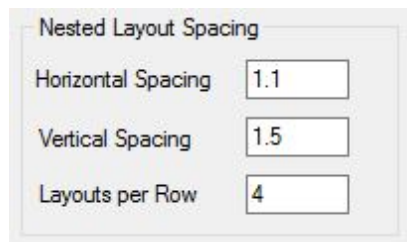
The default setting is Decimal Imperial.

Display Repeated Layouts

Using Display Repeated Layouts will show a nested sheet in the nest drawing for every sheet needed in the job. Typically, if there is a nest that needs to be run more than once, the quantity field will simply show the quantity of that sheet that you need to run, instead of making duplicate nests. So, if you run a job and the nest drawing shows 2 sheets, but there are several programs to run, then you can check the sheet quantity in the nested sheet for how many instances of that sheet are needed.

Nested Layout Spacing

The options in Nested Layout Presentation will allow you to format the result nest drawing so that the nested sheets are easier to see, read and print. You are allowed control over a system of layout that has rows and columns, referred to as Horizontal Spacing, Vertical Spacing, and Layouts per Row.



Horizontal Spacing

This is the setting for the horizontal spacing between the nested sheets when repeated horizontally. This would be similar to the spacing between Columns on a spreadsheet.

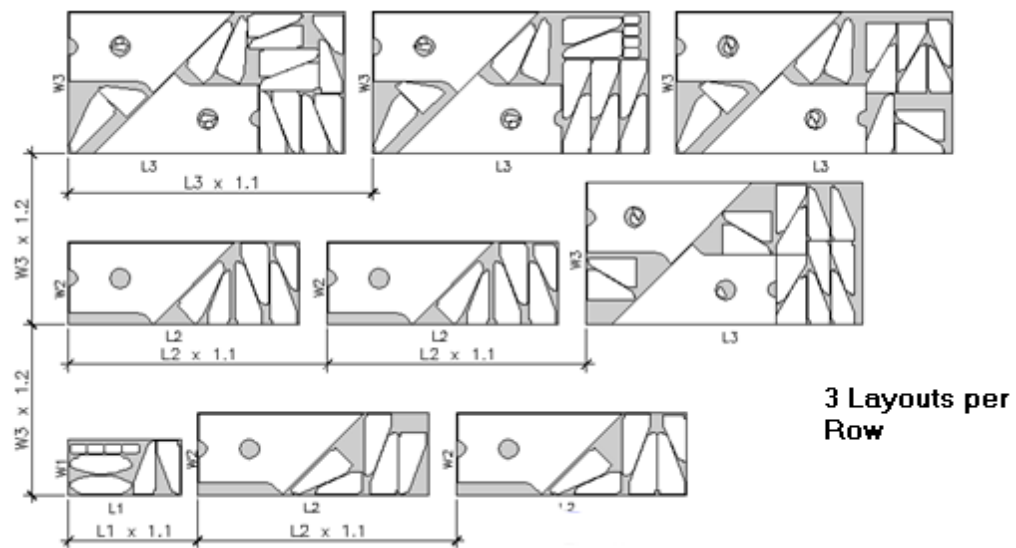
Vertical Spacing

This is the setting for the vertical spacing between the nested sheets when repeated vertically. This feature would be similar to the spacing between Rows in a spreadsheet.

NOTE: If you are using Automated Two-Side nesting, this value will need to be changed to 2.5 or greater.

Layouts per Row

This is the number of nested sheets in a row to display horizontally. The sheets will continue vertically until all sheets are shown.



Save Nests or Parts as Drawings

Checking this parameter will save a copy of the nest layout drawing or single parts drawings in the job results folder. This is useful to review the nest results for a job.

Save AutoNEST Parts and Tasks

Checking this option will create a folder inside the job results folder containing the AutoNEST parts and task files for the job. This is useful if you want to keep the results folders for backup and need to run the nest later.

Packing Direction Control

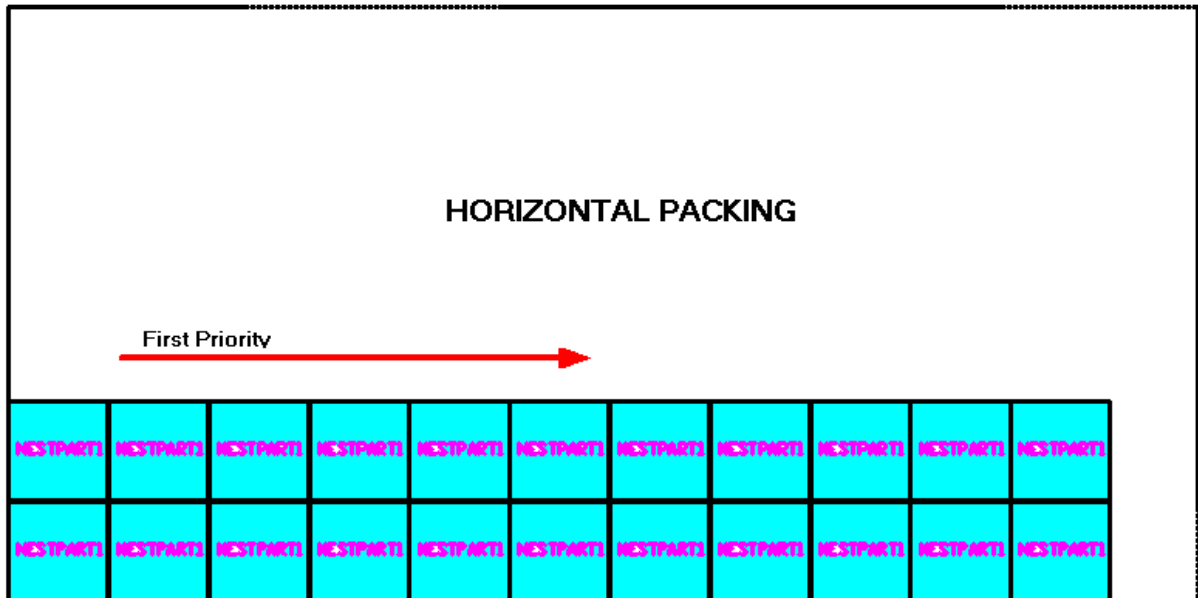
Packing Direction Control is the setting for the method used to place parts on the nested sheet.

Auto

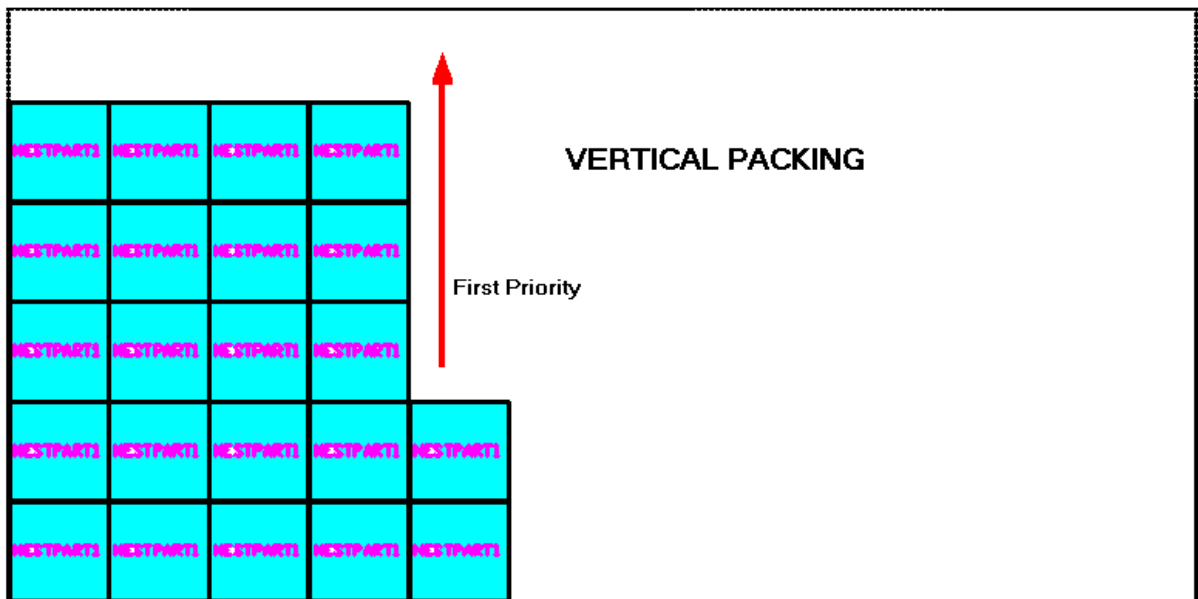
Auto will place parts with no particular preference for Horizontal or Vertical packing, but using the most efficient method to gain yield from the sheet.

Horizontal Packing

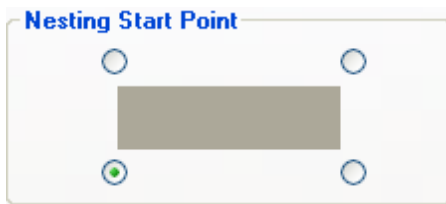
Horizontal Packing will favor placing parts across the sheet (in X) before moving up the sheet (in Y).

**Vertical Packing**

Vertical Packing will favor placing parts moving up the sheet (in Y) before moving across the sheet (in X).

**Nesting Start point**

This selection will change the location where nest starts placing the parts.



Sheet Origin

The sheet origin is the corner where 0,0 is located, affecting the creation of the NC code by generating X, and Y values relative to the start point.



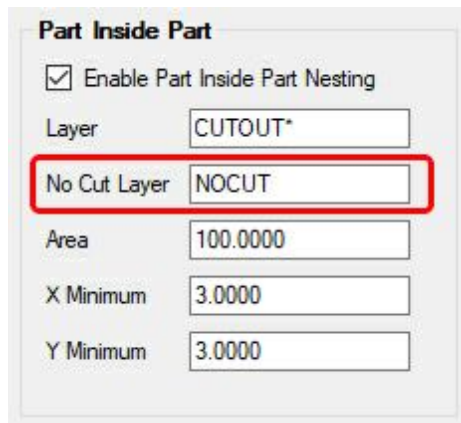
Part Inside of Part (PIP) Nesting

Part Inside of Part Nesting will allow you to nest parts within areas of other parts.

Part Inside Part	
<input checked="" type="checkbox"/>	Enable Part Inside Part Nesting
Layer	CUTOUT*
No Cut Layer	
Area	100.0000
X Minimum	3.0000
Y Minimum	3.0000

Layer - This is the layer name you want to use for parts to nest inside. For instance with Solid-CIM 3D this would be CUTOUT*. This name MUST be in ALL CAPS.

No Cut Layer - This is the layer name you want to use for parts to nest inside when this layer is not actually cut in the nest.



Part Inside Part

☒ Enable Part Inside Part Nesting

Layer: CUTOUT*

No Cut Layer: NOCUT

Area: 100.0000

X Minimum: 3.0000

Y Minimum: 3.0000

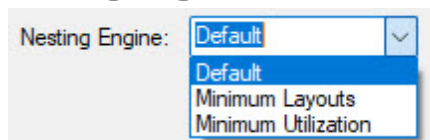
Area - This is the minimum area the geometry must be in order for nest to place parts on it. So, all cutouts that have at least this much area will be considered for PIP nesting.

X Minimum - The minimum size in X for a part to be considered for PIP nesting. For instance if a cutout is bigger than area but it is only 3" long by 40" tall with X Minimum set to 5.00 then this cutout will not get parts nested inside of it because it is not at least 5.00 in the X dimension.

Y Minimum - The minimum size in Y for a part to be considered for PIP nesting. For instance if a cutout is bigger than area but it is only 3" wide by 60" long with Y minimum set to 5.00 then this cutout will not get parts nested inside of it because it is not at least 5.00 in the Y dimension.

If you create a layer on a part that includes _PIP as the suffix, this will trigger the nesting engine to see if it can nest any parts within the geometry that is on that layer as well.

Nesting Engine



Nesting Engine: Default

- Default
- Minimum Layouts
- Minimum Utilization

The default is NESTPRO which is best suited for most jobs.

"Minimum Layouts" engine is only suitable when the stock materials are inexpensive and the objective is to maximize the repeatability of the nested layouts or cutting plan.

However when the parts' quantities are high, it is recommended you use "Minimum Utilization" nesting engine. This engine is designed to optimize the utilization of the stocks as well as to achieve a high repeatability of the layouts, thus minimizing the number of different nested layouts.

Advanced Nesting Module

Advanced nesting is a set of features that allow for more efficient methods of cutting parts in a nest. There are methods to keep the tool down in the material for as long as possible while cutting all the

profiles of the parts (Staydown). There is also a method to cut in on two parts at the same time by cutting between the parts and spacing the parts so that the cutting tool contacts a part on each side as it moves through the sheet (Common Line). One advanced method involves cutting both sides of a sheet while the parts are still nested together by first cutting one side, then flipping the sheet over and cutting the other side, cutting the parts loose at that time (Backside Nesting). The final advanced method is a procedure that allows you mix multiple parts together, but control the number of parts defined by priority that get mixed into one sheet, allowing you to sort the parts easier as they come off the machine, but still allowing higher yields in the sheet by allowing the maximum number of parts/sizes available to fill the nest (Cart Control Nesting).

There are additional advanced nesting controls located in the [Material Database](#) section including [Multi-Stock Nesting](#) and [Skeleton Scrap Cut](#).

Each of these methods are described here along with their required settings.

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
<div> <div> Nesting Bridge Parameters <input checked="" type="radio"/> Off <input type="radio"/> Staydown <input type="radio"/> Commonline Bridge Width: -0.1250000000 Minimum Corner Offset: 0.5000000000 Minimum Bridge Angle: 50.0000000000 Maximum Bridges Per Part: 4 Bridge Override: <input type="checkbox"/> Enable Square Corners for Staydown Bridge Radius: 0.0000 </div> <div> Veneer Match Parameters <input checked="" type="radio"/> Display Individual Parts <input type="radio"/> Display Outline of Veneer Match <input type="radio"/> Display Parts and Outline of Veneer Match Veneer Match Bridge: 0.0000 </div> <div> Skeleton Cutting <input checked="" type="radio"/> Entire Sheet <input type="radio"/> Enclosing Rectangle Minimum X: 25.0000 Minimum Y: 25.0000 </div> </div> <div> <div> Backside Nesting <input type="checkbox"/> Enable Backside Nesting <input type="checkbox"/> Entire Width of Sheet Pin Trim Amount Y: 18.0000 <input checked="" type="checkbox"/> Entire Length of Sheet Pin Trim Amount X: EDGE </div> <div> Small Part Zones <input checked="" type="radio"/> Entire Part Inside <input type="radio"/> Center of Gravity Inside Zone Extension Allowed (%): 0.00 Small Part Edge Offset: 0.0000 </div> <div> Staydown and Part Inside Part Bridging <input checked="" type="radio"/> Bridge Parts to One Another and PIP Layer <input type="radio"/> Bridge Parts to One Another Only <input type="radio"/> Disable Bridging For Part Inside Part Open Cart Control Grouping Key: None </div> </div>								

Nesting Bridge Parameters

Off

Nesting Bridge Parameters

☒ Off ☐ Staydown ☐ Commonline

Bridge Width: -0.1250000000

Minimum Corner Offset: 0.5000000000

Minimum Bridge Angle: 50.0000000000

Maximum Bridges Per Part: 4

Bridge Override:

☐ Enable Square Corners for Staydown

Bridge Radius: 0.0000

Setting this parameter to '**Off**' will create a standard type of nest and place the parts on the sheet with all their tool paths. This is the default method of nesting with no advanced features. Typically this results in a separate tool path around each, individual part in the nest.

Staydown Nesting

Nesting Bridge Parameters

☐ Off ☒ Staydown ☐ Commonline

Bridge Width: -0.1250000000

Minimum Corner Offset: 0.5000000000

Minimum Bridge Angle: 50.0000000000

Maximum Bridges Per Part: 4

Bridge Override:

☐ Enable Square Corners for Staydown

Bridge Radius: 0.0000

Staydown nesting will attempt to place the tool in the nest and leave the tool in the material as much as possible as it cuts around all the parts. This will typically create small bridges between the parts as the tool cuts around them.

Bridge Width

Use this parameter to set the width of the bridges between the parts. These bridges can be a positive value so that there is a tab holding the parts together, or it can be a negative value so that the tool overlaps the starting point of the bridges.

Note: If you are using **STAYDOWN** with cutter compensation set to **YES**

1) You will need a bridge width set to the **DIAMETER** of the tool you are using plus the **OVERLAP** you are looking for. This should be a negative number.

2) Bridge Override needs to be set to 0.0

Minimum Corner Offset

The minimum corner offset is how far from the corner of the nested part the bridge can appear. You do not want the bridge on the very corner of the part, especially if you wish to use a negative tab that the tool will cut off. Typically you would put a number in there that would allow for the entire tool diameter to move from the tab to the corner of the part (so at least 1 tool diameter plus).

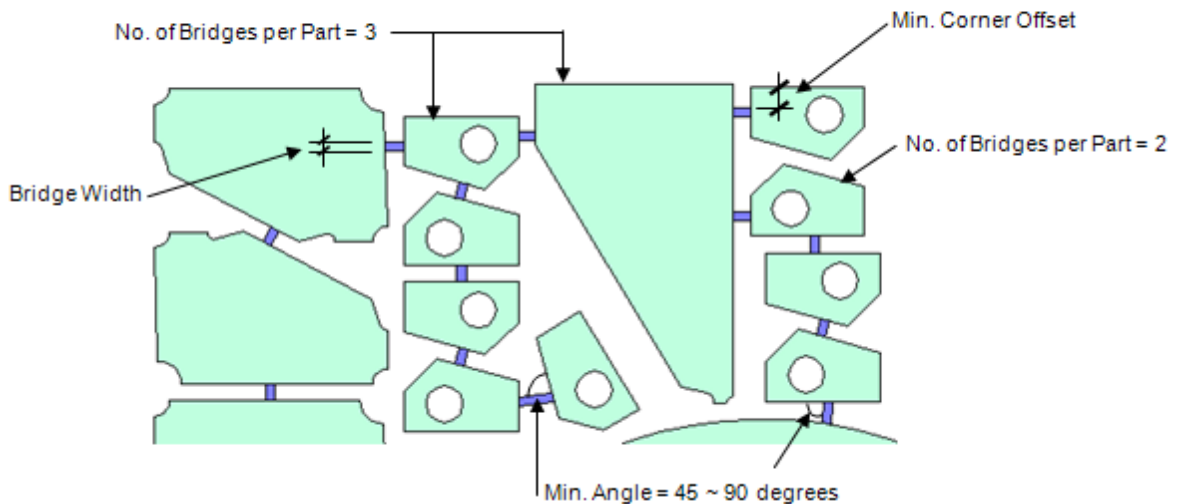
Minimum Bridge Angle

This is an angle between 45°-90°, to specify the minimum angle to consider for placing a bridge between two parts. Typically you would not want an angle that was too acute, or the bridge and resulting corner can become unsuitable for a tool to cut.

Maximum Bridges Per Part

There will often need to be several bridges on a part depending on the nest. This field limits the number of bridges that the nest can use to hold the parts together. This sometimes means that there will be more tool paths made if the number of bridges is too small. A good starting default is 4.

Illustration for Bridge Parameters



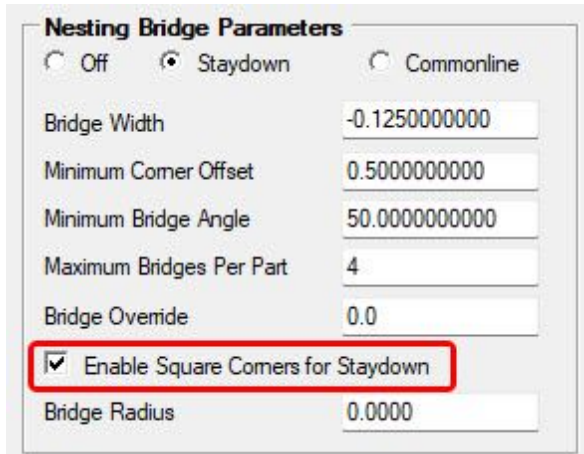
Bridge Override

This field defines the offset of the STAYDOWN cut path. This field should be left blank if you are using Cutter Compensation in your STAYDOWN cut knowledge set to NO or BOTH. If the Cutter Compensation in your STAYDOWN cut knowledge is set to YES, this should be set to 0.0.

Nesting Bridge Parameters		
<input type="radio"/> Off	<input checked="" type="radio"/> Staydown	<input type="radio"/> Commonline
Bridge Width	-0.1250000000	
Minimum Corner Offset	0.5000000000	
Minimum Bridge Angle	50.0000000000	
Maximum Bridges Per Part	4	
Bridge Override	0.0	
<input type="checkbox"/> Enable Square Corners for Staydown		
Bridge Radius	0.0000	

Enable Square Corners for Staydown

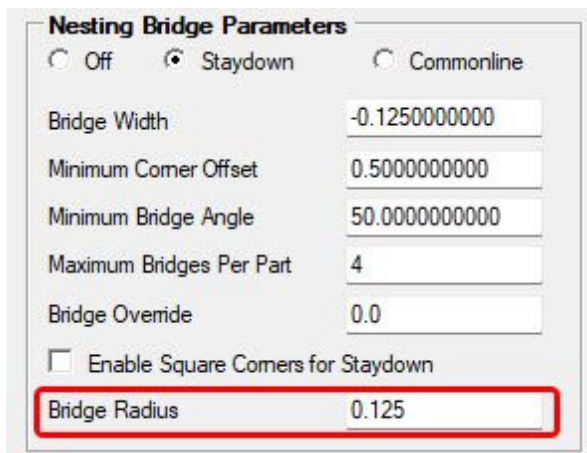
The STAYDOWN geometry created by Router-CIM Automation Suite has the rounded corner function in use by default. In order to change this function, select this option.



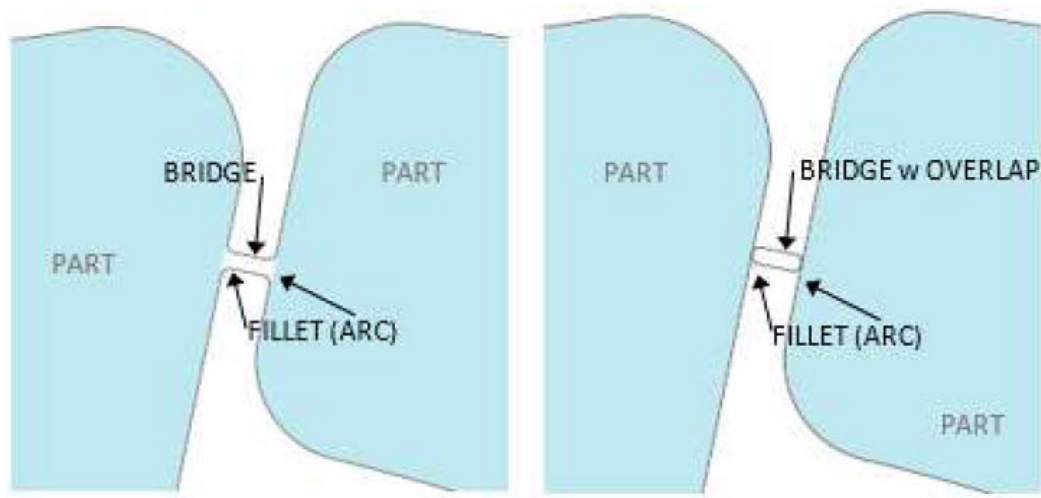
The screenshot shows the 'Nesting Bridge Parameters' dialog box. At the top, there are three radio buttons: 'Off', 'Staydown' (which is selected), and 'Commonline'. Below these are several input fields: 'Bridge Width' with a value of -0.1250000000, 'Minimum Corner Offset' with 0.5000000000, 'Minimum Bridge Angle' with 50.0000000000, 'Maximum Bridges Per Part' with 4, and 'Bridge Override' with 0.0. A checkbox labeled 'Enable Square Corners for Staydown' is checked and highlighted with a red rectangle. At the bottom, the 'Bridge Radius' field has a value of 0.0000.

Bridge Radius

The STAYDOWN geometry created by Router-CIM Automation Suite can use an arc bridge when creating the bridge from part to part. Enter the radius of the bridge that is less than the diameter of the tool being used.



This screenshot shows the same 'Nesting Bridge Parameters' dialog box. The 'Staydown' radio button remains selected. In this view, the 'Enable Square Corners for Staydown' checkbox is unchecked. The 'Bridge Radius' field at the bottom is highlighted with a red rectangle and contains the value 0.125.



Creating the STAYDOWN Cut Knowledge

The Cut cycle should be **Heli-Lead Center** with no offsetting.

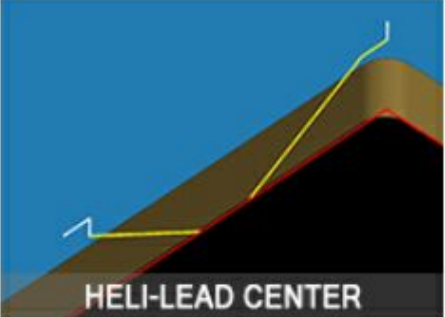
Note: In your DOIT file, do not cut the outside part layer. Turn that association off.

The DOIT file needs a knowledge to layer association to cut the tool stay down geometry created by AutoNest. You must make a knowledge called **STAYDOWN** and associate it to a layer called **STAYDOWN**. This one knowledge will be used to cut all materials in your job.

Cutter Compensation YES, NO or BOTH (Refer to the Cycle Information Column highlighted):

The Staydown knowledge can use cutter compensation with the YES, NO or BOTH. No changes need to be made when using NO or BOTH under cutter compensation. When using cutter compensation set to YES, make sure Bridge Override is set to 0.0 for an accurate cut path. This will allow the machine to offset the STAYDOWN tool path.

You will need to adjust the Cut Side parameter to '**Outside**' as shown below:

Cycle Information		Status Information	
Offset Dim	0.0	Safety Plane	*0.25000
Cut Side	OUTSIDE	Depth Per Pass	1.00000
Cut Direction	CW	Total Cut Depth	M/- .005
Round Corners	n	Feedrate/Spindle Speed	
Lead Size	0.0	Feedrate	1000.00000
Lead Angle	1.0	Spindle Speed	18000.00000
Lead Ratio		Surface FPM	NONE
Lead Feed		Units Per Revolution	NONE
Overlap Amount	AUTO	Calculate	
XY Stock Allowance		Before Codes	
Z Stock Allowance		After Codes	
		Oscillation Amount	0.00000
		Sort By Rank #	600
			
		Reset Cycle Settings to Default	

If you need a different knowledge for each material in your job, you can make a layer / knowledge association formatted like this:

Knowledge name: **MATERIAL CODE+STAYDOWN**

Layer name: **MATERIAL CODE+STAYDOWN**

As an example, The ¾ MDF material has a material code of 75MDF and parts on this material will use a knowledge called 75MDFSTAYDOWN and their staydown geometry will be on a layer called 75MDFSTAYDOWN. The tool radius in this knowledge is used as the offset for part nesting. Other knowledges can cut this layer as well but the stay down geometry can only be cut on center and cannot be offset.

If you have parts on other materials, they will be cut using the regular STAYDOWN knowledge because they do not have a knowledge associated with the material code.

Small part cutting can still be used by making a knowledge that cuts the outside geometry and has a lower rank than the STAYDOWN knowledge. The small part cutting knowledge would be cutting the SML outside part layer. See ['Rename Outside Layers'](#) section.

Common Line Nesting

Nesting Bridge Parameters

☐ Off
 ☐ Staydown
 ☒ Commonline

Bridge Width: -0.1250000000
 Minimum Corner Offset: 0.5000000000
 Minimum Bridge Angle: 50.0000000000
 Maximum Bridges Per Part: 4
 Bridge Override:
☐ Enable Square Corners for Staydown

Common Line will create a series of tool paths between the parts attempting to make the fewest tool path moves and still cutting all the parts. This means that the tool will be cutting parts on both sides of the tool path, instead of making one tool path around each individual part. **Common Line nesting is designed for rectangular parts only.**

The Cut cycle should be **Center Line Cut or Center Line Ramp** with no offsetting.

Note: In your DOIT file, do not cut the outside part layer. Turn that association off.

The DOIT file needs a knowledge to layer association to cut the common line geometry created by AutoNest. You must make a knowledge called **COMMONLINE** and associate it to a layer called **COMMONLINE**. This one knowledge will be used to cut all materials in your job.

If you need a different knowledge for each material in your job, you can make a layer / knowledge association formatted like this:

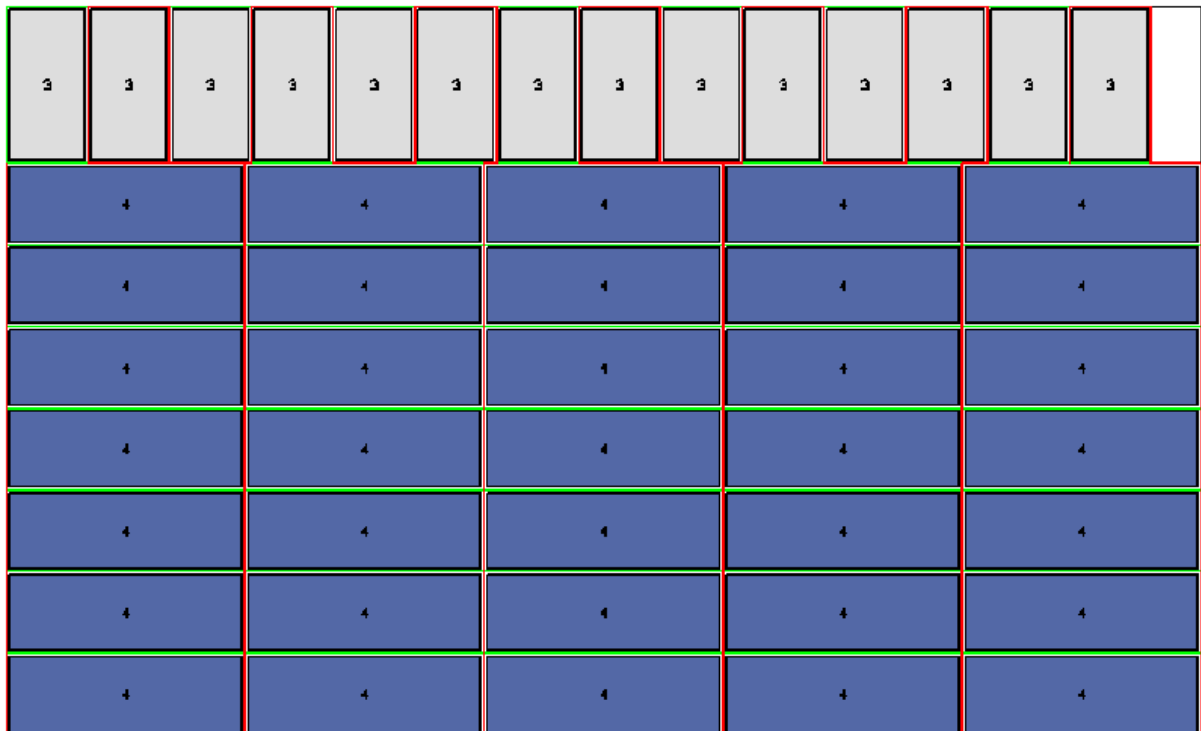
Knowledge name: **MATERIAL CODE+COMMONLINE**

Layer name: **MATERIAL CODE+COMMONLINE**

As an example, The ¾ MDF material has a material code of MDF75 and parts on this material will use a knowledge called MDF75COMMONLINE and their common line geometry will be on a layer called MDF75COMMONLINE. The tool radius in this knowledge is used as the offset for part nesting. Other knowledges can cut this layer as well but the common line geometry can only be cut on center and cannot be offset.

Parts in the same job on different materials will need a knowledge / layer association that matches the MATERIAL CODE+COMMONLINE format.

Consider the following nest of parts, cut with Common Line:



© 2025 CIM-TECH.COM, Inc.

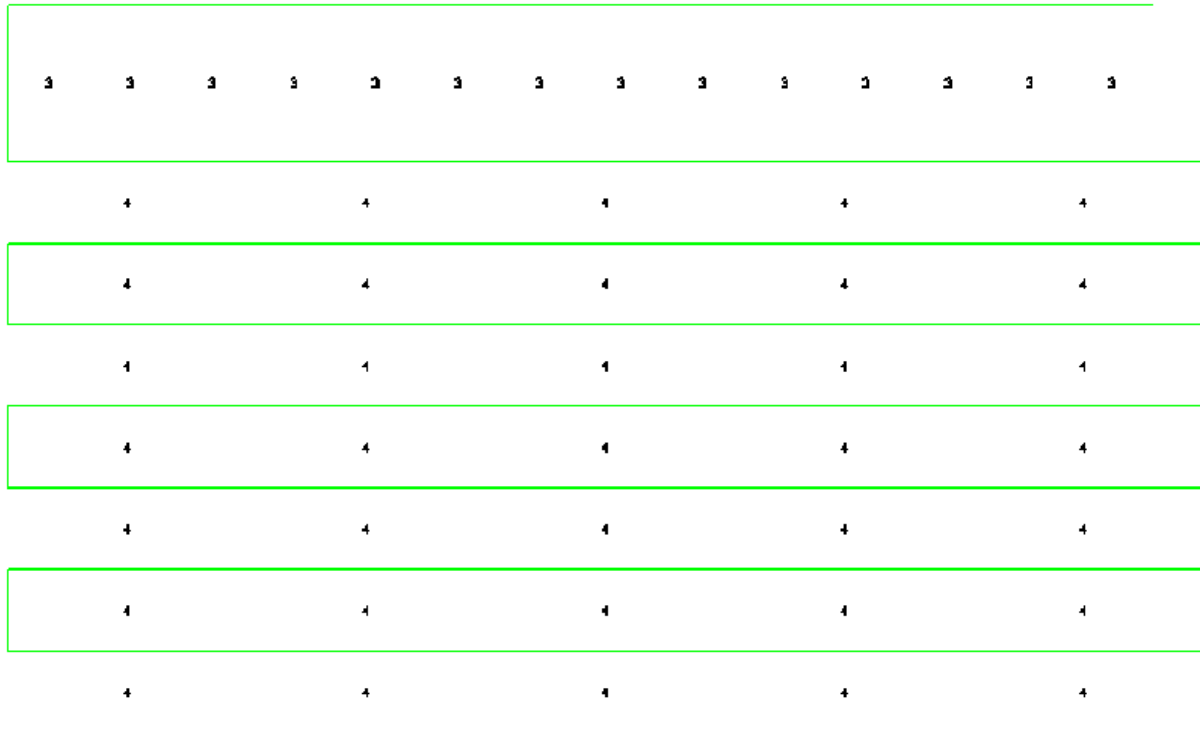
3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4

Here the hint of fact that there are really only two tool paths on the entire nest starts to show. If we isolate just the vertical tool path (red line shown) we can see:

3	3	3	3	3	3	3	3	3	3	3	3	3	3
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4
4	4	4	4	4	4	4	4	4	4	4	4	4	4

The above illustration shows that the tool path winds through the parts without lifting, cutting each of the vertical sections as it moves through the nest.

Further isolating the horizontal tool paths shows the rest of the tool path.



This is the horizontal path that cuts the rest of the geometry in the nest.

The end result being that with only two tool paths the tool is able to stay in the nest longer, eliminating the constant index moves from part to part and also is better able to maintain a more constant chip load, which keeps the tool cooler and makes it last longer. The result being a faster and more efficient process.

Automated Two-Sided Nesting

Backside Nesting

☒ Enable Backside Nesting

☐ Entire Width of Sheet

Pin Trim Amount Y

☒ Entire Length of Sheet

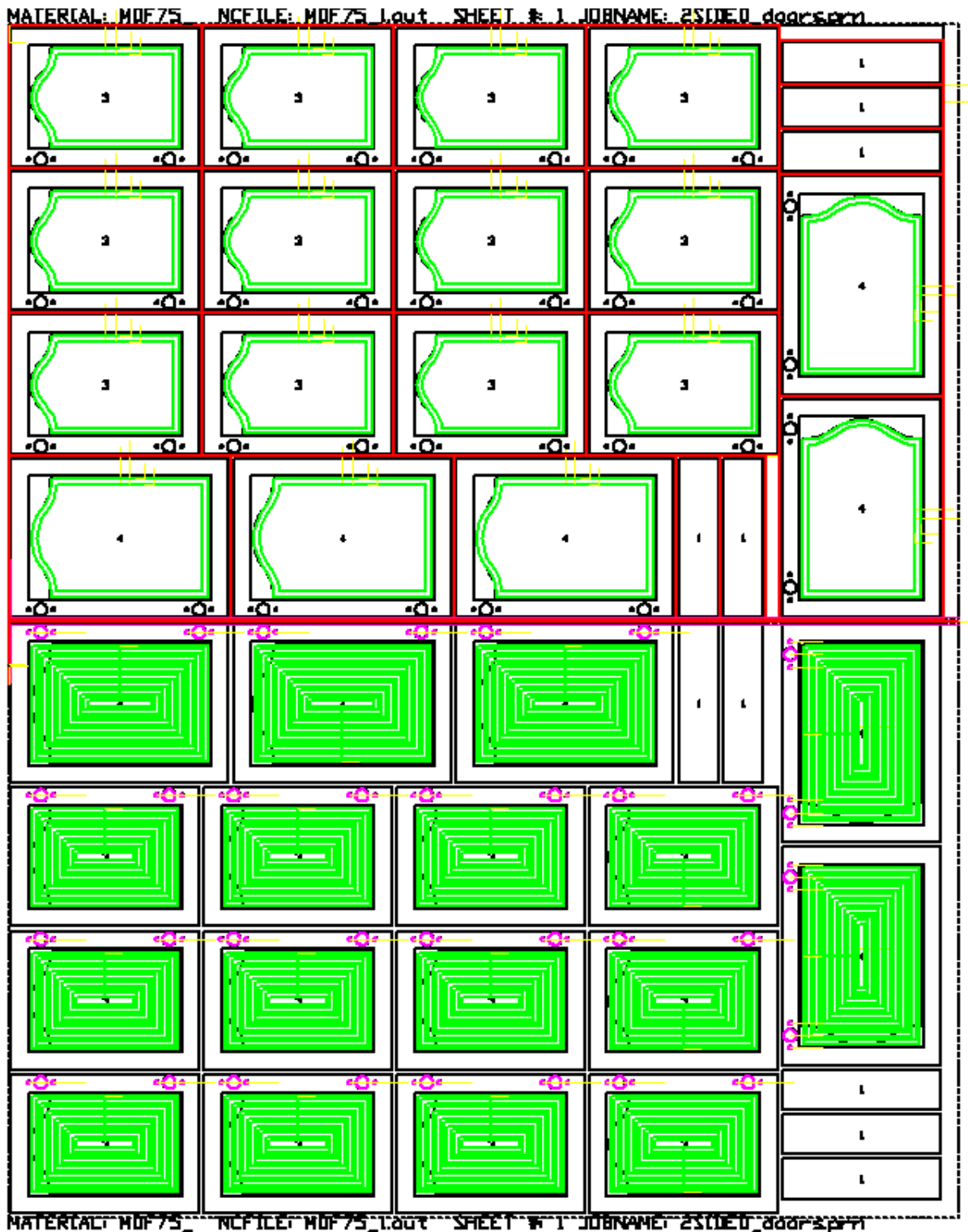
Pin Trim Amount X

Use this option to turn on and off the Automated Two-Side Nesting feature.

Automated Two-Side Nesting will allow you to cut on both sides of a nested sheet, enabling you to make all the necessary router operations on a part without having to place any parts back on the machine one at a time for re-processing.

Typically the backside is cut first. Outside profile cuts (through cuts) can be made halfway through on the back and then all the way through on the front, or they can be cut all the way through on the front side (second side).

Consider the following nest of parts:

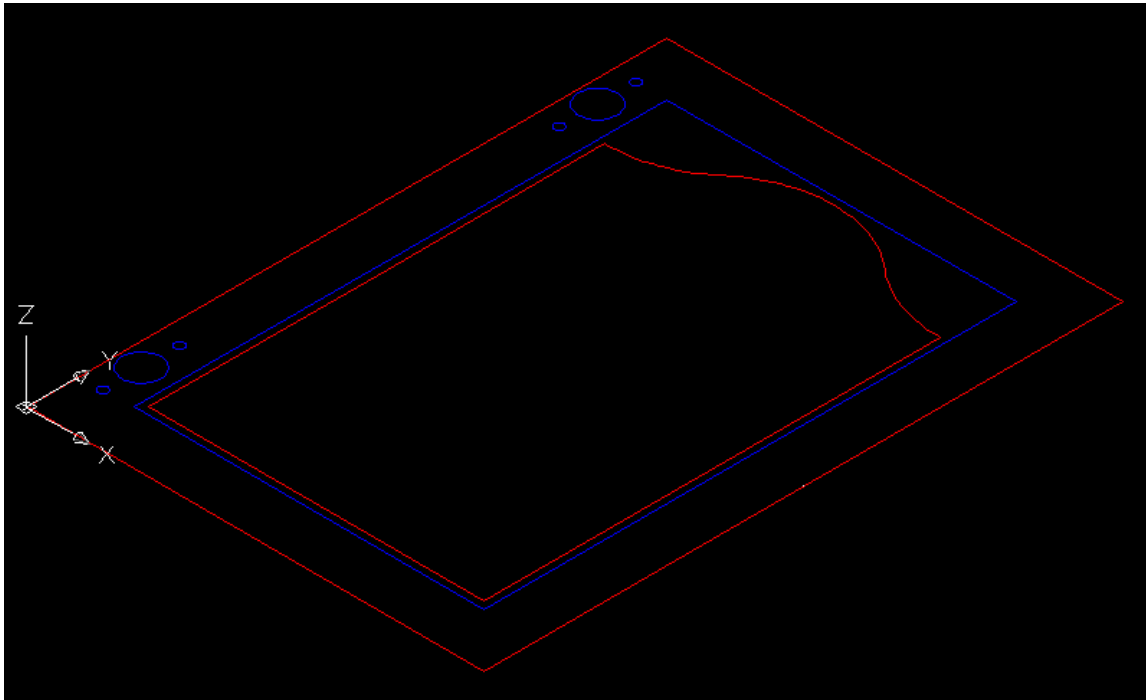


In this scenario, the bottom sheet has all the glass pockets, hinge hardware cuts, etc. that will be necessary on the backside. Then there is a reference cut made on the top and top left corner of the sheet. Next the sheet can be flipped over with the reference point being placed against the location pins. Then all the parts can have the details cut and then the profiles can be cut, releasing the part from the

sheet. In this case, Common Line cutting could also be used on the top to reduce the cycle time necessary to release the parts from the sheet.

Operation

The general rules to this method are that all the geometry for the part (front and backside) must be in one drawing and on separate layers that can be controlled with a layer to knowledge association in the DOIT files.

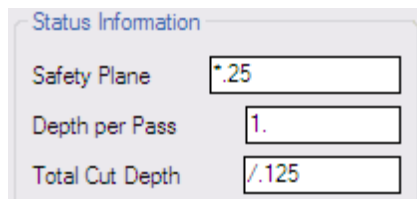


In the drawing shown above, the red geometry is cut on the front side of the sheet and the blue geometry is cut on the backside of the sheet. Both front and back geometry must exist in the same drawing, DXF, or macro file.

The front and or back geometry can have negative thickness. If you want your knowledge to cut to the geometry thickness, then your total cut depth can either be left blank, or have a capital A, or a backslash and a value. Blank or A will make the cut depth to the geometry thickness. A backslash and a value will cut that value more or less than the geometry thickness:

Status Information	
Safety Plane	<input type="text" value="-.25"/>
Depth per Pass	<input type="text" value="1."/>
Total Cut Depth	<input type="text" value="/-.005"/>

This example will cut .005 MORE than the geometry thickness. If the geometry is -.75 thick, the cut depth will be -.755



Status Information	
Safety Plane	.25
Depth per Pass	1.
Total Cut Depth	/.125

This example will cut .125 LESS than the geometry thickness. If the geometry is -.75 thick, the cut depth will be -.625

In the DOIT editor, there are two tabs. One for cuts on the first side and one for cuts on the second side. The order of these are as follows; The **Backside** tab is for the cuts that will happen first, before the sheet is turned over. The **Front Side** tab is for the cuts that will happen when the sheet is flipped over, containing all the cuts that will happen last and when the parts are cut loose from the sheet.

Knowledge to Layer Association

Front Side Back Side

Turn Off	Knowledge	Layer
<input type="checkbox"/>	PIN_TRIM	PIN_TRIM
<input type="checkbox"/>	PKTCUTOUT_FINISH	CUTOUT_FRAME
<input type="checkbox"/>	PKTCUTOUT_FINISH	PKTCUTOUTBOTTOM_...
<input type="checkbox"/>	PKTCUTOUT_ROUGH	CUTOUT_FRAME
<input type="checkbox"/>	PKTCUTOUT_ROUGH	PKTCUTOUTBOTTOM_...
<input type="checkbox"/>	PKHINGE_250	HOLE_HINGE_1

Knowledge to Layer Association

Front Side Back Side

Turn Off	Knowledge	Layer
<input checked="" type="checkbox"/>	_375_DRILL_FULL	_375_DRILL_FULL
<input type="checkbox"/>	_45CUT_FULL	_45CUT_FULL
<input type="checkbox"/>	_INSIDE_CUT	_INSIDE_CUT
<input type="checkbox"/>	_OUTSIDE_CUT	_OUTSIDE_CUT
<input type="checkbox"/>	_OUTSIDE_CUT_SML	SML_OUTSIDE_CUT
<input type="checkbox"/>	_QUART_DRILL_FULL	_QUART_DRILL_FULL
<input type="checkbox"/>	_TNUT_DRILL_FULL	_TNUT_DRILL_FULL
<input type="checkbox"/>	BLANK	BLANK
<input type="checkbox"/>	BORDER	BORDER
<input type="checkbox"/>	BORE_250	BORE
<input type="checkbox"/>	COR_INSIDE	COR-INSIDE
<input type="checkbox"/>	COR-DRAIN	COR-DRAIN
<input type="checkbox"/>	COR-ENGRAVING	COR-ENGRAVING
<input type="checkbox"/>	COR-OUTSIDE	COR-OUTSIDE
<input type="checkbox"/>	CUTOUT_WAINSCOT	CUTOUT_WAINSCOT
<input type="checkbox"/>	CUTOUT_WAINSCOT	PROFILE_RPJAMB
<input type="checkbox"/>	DADO	CUTOUTBACKDADO
<input type="checkbox"/>	DADO	CUTOUTBOTTOM

If you want to cut half way through on the front, and half way through on the back, then you need a knowledge and layer association that cuts half way through for both the Normal tab and another (or the same) knowledge association for the After Nest tab. If you want to cut all the way through on the front side, then that knowledge / layer association should be done in the Normal machining tab.

Note: You would never cut all the way through on the Backside because that would separate all the parts and you could not flip the sheet over!

Once you make the associations in these two lists, save the DOIT file.

The Automated Two-Side Nesting is activated by checking the box in the Advanced Nesting tab that says '**Enable Backside Nesting**'.

Backside Nesting

☒ **Enable Backside Nesting**

☐ Entire Width of Sheet

Pin Trim Amount Y

☒ Entire Length of Sheet

Pin Trim Amount X

The vertical spacing needs to be set at 2.5 or more. This ensures that there is enough room to mirror the sheets.

For more information on changing the vertical spacing, refer to the [Nesting](#) section.

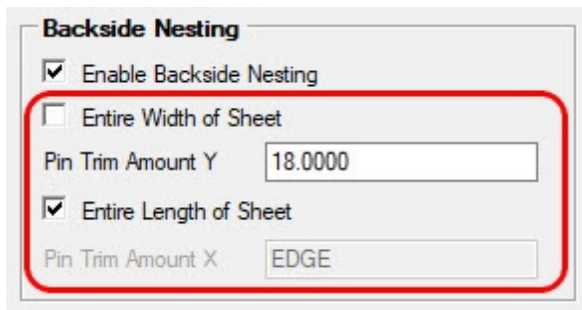
Note that Automated Two-Side Nesting requires rectangular stock only, irregular stock shapes with more than 4 sides will not be mirrored.

Pin Trim Cut

A trim cut is made along the edge/edges of the backside sheet. This trim cut ensures a good edge to place up on the pop up pins when the sheet gets flipped over.

Router-CIM Automation Suite automatically makes geometry on layer **pin_trim**. You need a knowledge named **pin_trim** associated to layer **pin_trim** in the backside [DOIT](#) file.

The pin trim geometry will be all the way on the X axis by default and 18.0 units in the Y axis. This can be adjusted by changing the 'Pin Trim Amount' value. Both the X and Y axes pin trim geometry can be adjusted by either checking or unchecking the 'Entire Width of Sheet' or 'Entire Length of Sheet'. If unchecked, you will need to define the amount of units to make the length of the respective pin trim cut.



Backside Nesting

☒ Enable Backside Nesting

☐ Entire Width of Sheet

Pin Trim Amount Y: 18.0000

☒ Entire Length of Sheet

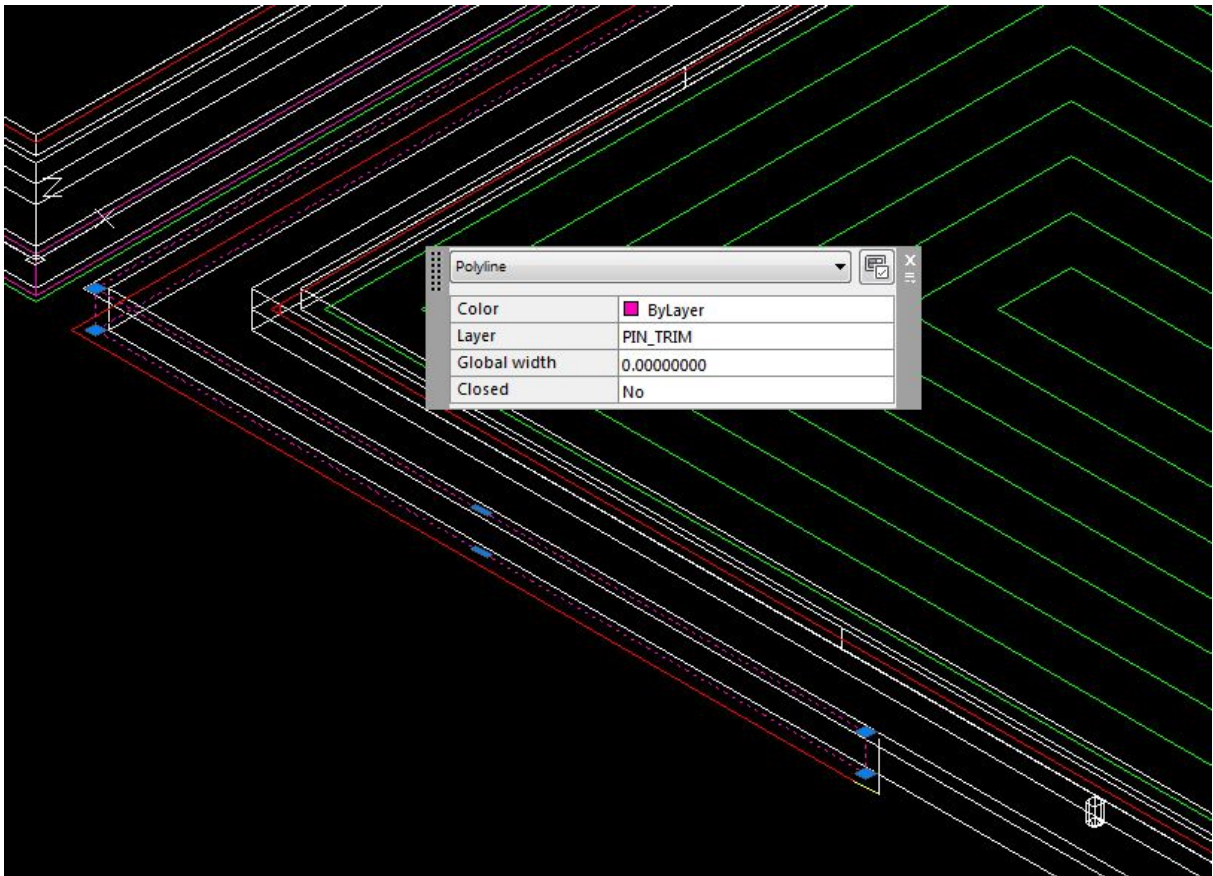
Pin Trim Amount X: EDGE

If you need a different knowledge for each material in your job, you can make a layer / knowledge association formatted like this:

Knowledge name: **MATERIAL CODE+PIN_TRIM**

Layer name: **MATERIAL CODE+PIN_TRIM**

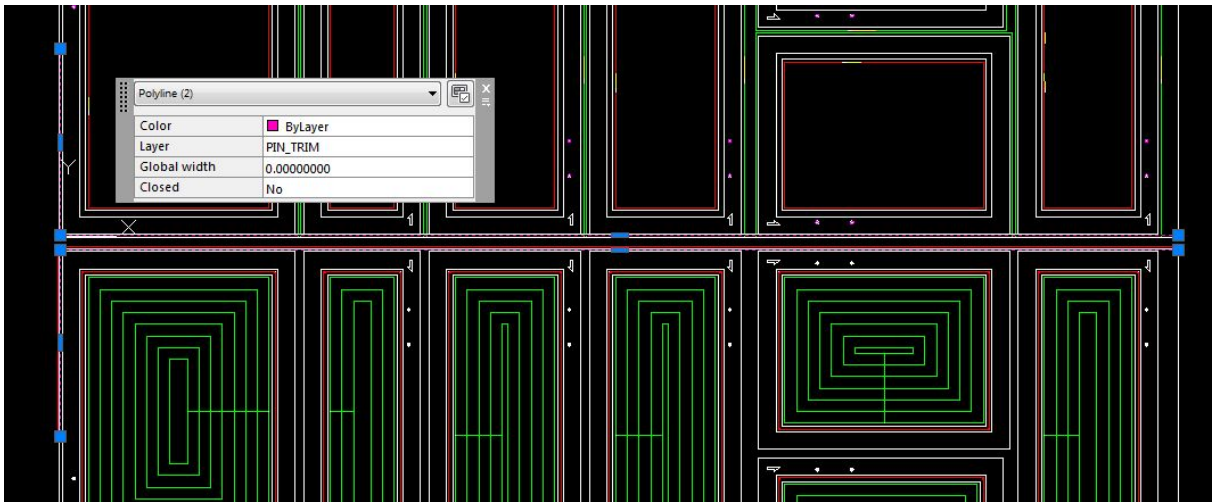
As an example, The ¾ MDF material has a material code of 75MDF and the pin trim cut on this material will use a knowledge called 75MDFPIN_TRIM and the pin trim geometry will be on a layer called 75MDFPIN_TRIM.



Note: pin_trim Knowledge: This will be a Right-Hand (RH) Cut set to offset by the tool radius, cut to the RH and CCW direction. This is based on lower left Sheet Origin being selected. A different sheet origin may require a different cycle such as Left-Hand (LH) Cut.

The **pin_trim** geometry is offset by $\frac{1}{2}$ of the edge allowance and will be shaved off the backside of the sheet and the origin of the main NC code file will be shifted by that amount. To change the default offset, please go to the ['Common NCVARS in Automation'](#) section.

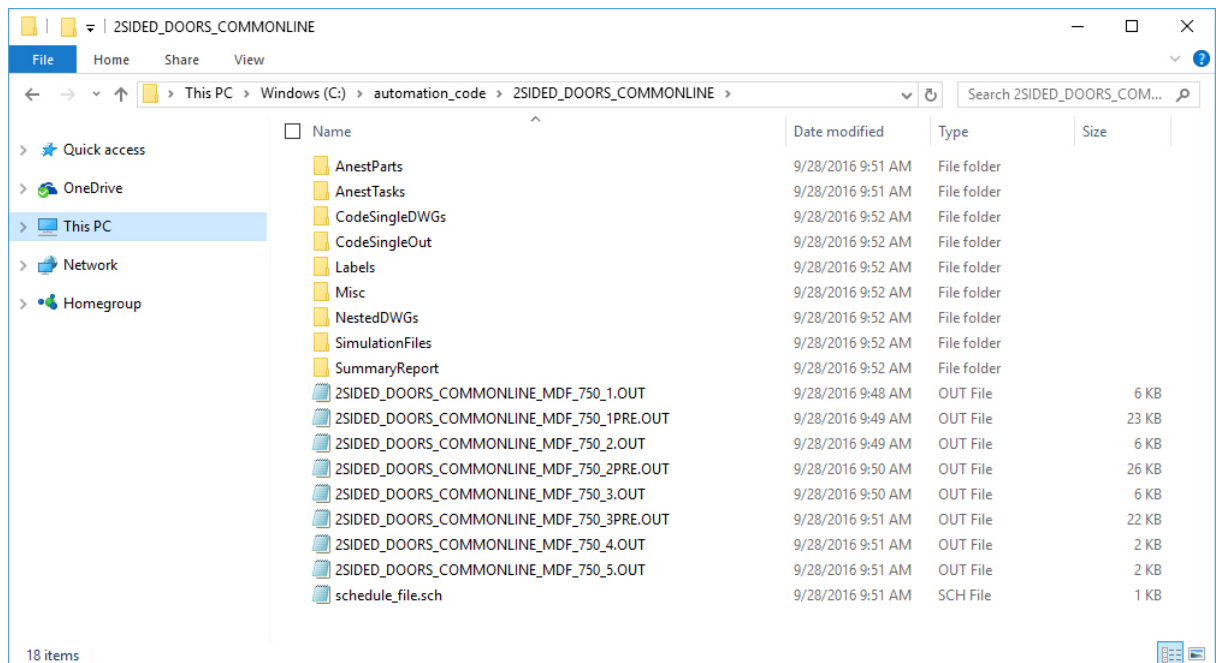
The spacing between the front side nest and backside nest is defaulted to the bottom edge allowance defined in the material. If you want to adjust the spacing between the front side nest and backside nest in the nested DWG that is produced, please go to the ['Common NCVARS in Automation'](#) section.



Results

When the job is run, the results will have two NC Code files for each sheet.

The regular sheet name and one with PRE in the name. The PRE is the backside and it is run first.



Schedule file

The schedule file will be created with the files listed in the proper order to be executed.

```
1,2SIDED_DOORS_COMMONLINE_MDF_750_1PRE.OUT,1,1,00:00:00,00:00:00
1,2SIDED_DOORS_COMMONLINE_MDF_750_1.OUT,1,1,00:00:00,00:00:00
2,2SIDED_DOORS_COMMONLINE_MDF_750_2PRE.OUT,1,1,00:00:00,00:00:00
2,2SIDED_DOORS_COMMONLINE_MDF_750_2.OUT,1,1,00:00:00,00:00:00
3,2SIDED_DOORS_COMMONLINE_MDF_750_3PRE.OUT,1,1,00:00:00,00:00:00
3,2SIDED_DOORS_COMMONLINE_MDF_750_3.OUT,1,1,00:00:00,00:00:00
4,2SIDED_DOORS_COMMONLINE_MDF_750_4.OUT,1,1,00:00:00,00:00:00
5,2SIDED_DOORS_COMMONLINE_MDF_750_5.OUT,1,1,00:00:00,00:00:00
```

Cart Control Nesting

Cart Control Nesting will allow you to control, to some degree, the parts that are nested together on your sheet stock in Router-CIM Automation Suite. Normally if you place parts together in Router-CIM Automation Suite, there is no control over which parts will end up together on a sheet, since yield is the highest concern at that point. This can lead to confusion or lost time as the parts are sorted at the machine, after being cut, so that they can be stacked together with other parts from the same priority.

As an example, if you had parts for 5 different assemblies together in one job, it is entirely possible to have parts from all 5 assemblies on the same sheet. This would cause the workers removing the parts to have 5 stacks (or carts) at the machine so that they can place the parts from the same assemblies together. A more efficient way of performing these tasks might be to limit how many carts (or stacks) can be nested together on the same sheet.

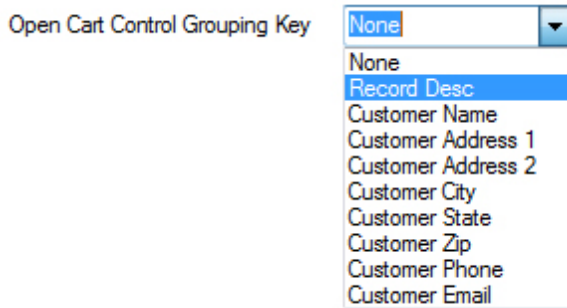
So, if the max number of carts is set to 2, then you can only have parts from 2 carts together on the sheet at one time until one of the two carts is finished, then another cart is added so that there are still only two until one of those carts runs out of parts, then another is substituted, and so on until all the parts from all 5 carts are finished. This way there are no more than two carts to stack at one time, or possibly 3 if you run out of parts for a cart part way through a sheet.

To control this in Router-CIM Automation Suite, you must have the Advanced Nesting option and under the Advanced Nesting tab set what field will be the **Open Cart Control Grouping Key** in the part parameters.

Open Cart Control Grouping Key

This is the field you can use to set the cart name or number for each part. You can choose from Record Description, or Label Fields 1-8 (Customer Info 1-8 by default).

Whichever key you choose, when adding parts to a job, you will place the cart name or number in the specified field so that the nest program can group the cart parts together.



Once this is set, in your job you should set which parts belong to which job (or cart). In the case below, the part is set for Cart 1. You can use any description for the field that identifies the cart, as long as all the parts in the same cart use exactly the same description.

Part Properties

Part Information Knowledge Information

Part Name: C:\vcim_work\Automation\Samples\DWGS\Sample1.dwg

Record Desc:

Material: MATERIAL_0.5000

Quantity: 4 ☐ Maximize parts on sheet of material

Filler Qty: 0

Label Information

Customer Info 1: ORD# 213423

Customer Info 2: Interior1

Customer Info 3:

Customer Info 4:

Customer Info 5:

Customer Info 6:

Customer Info 7:

Customer Info 8: 1

☐ Inherit Label Data from Job

Veneer Match Parameters

Name:

Location Point:

Rotation:

Part Options

☐ Ignore Layer Panel

☒ Start Point on longest side of part

☐ Manual Origin Part

☒ Nest and Code As Single Part

☐ Nest Part

☐ Code As Single Part

☐ Has an Associated Backside Macro

☐ Is an Irregular Stock Shape

☐ Use ShapeDone Part when possible

Macro Dimensions

Length:

Width:

Depth:

Part Orientation

☐ Rotate Part Rotate Angle: 90

☐ Mirror Part Mirror Axis: Horizontal

Nest Rotation: Same As Material

Save Close

In the material, you can choose the number of Jobs per cart. To locate this option, go to your Material Database and select the material you want to setup for Cart Control. When you open the material, select on the Advanced Nesting Parameters tab and the Cart Control Parameters will be on the right side. This is really the number of groups of parts that have the same record description per sheet in this material. The Open cart threshold is an override that says if there is at least the percentage shown of available space then go ahead and add another group of parts with the same description on this sheet. In this case, only two carts can be on the same sheet because there would have to be 95% of the space (empty sheet) left over in order to add a third cart to the sheet.

Scrap Management Advanced Nesting Parameters

Skeleton-Cut-Off Parameters

Enable Skeleton-Cut-Off ☐

Offset Distance: 0.0000

X Spacing: 0.0000

Y Spacing: 0.0000

Equal Spacing ☐

of X Spacings: 0.0000

of Y Spacings: 0.0000

Multi-Stock Nesting

☒ Priority ☐ Automatically Decrease Qty

☐ Best Yield

☐ Best Price

Alternate Sizes

+ Add Alternate Size - Delete Alternate Size

YDim	XDim	Qty	Priority	Cost
49.0000	121.0000	999	8	0.0000
61.0000	97.0000	999	10	0.0000
61.0000	121.0000	999	10	0.0000

Cart Control Parameters

Jobs per Cart: 2

Open Cart Threshold: 95

☒ Allow Veneer Matching

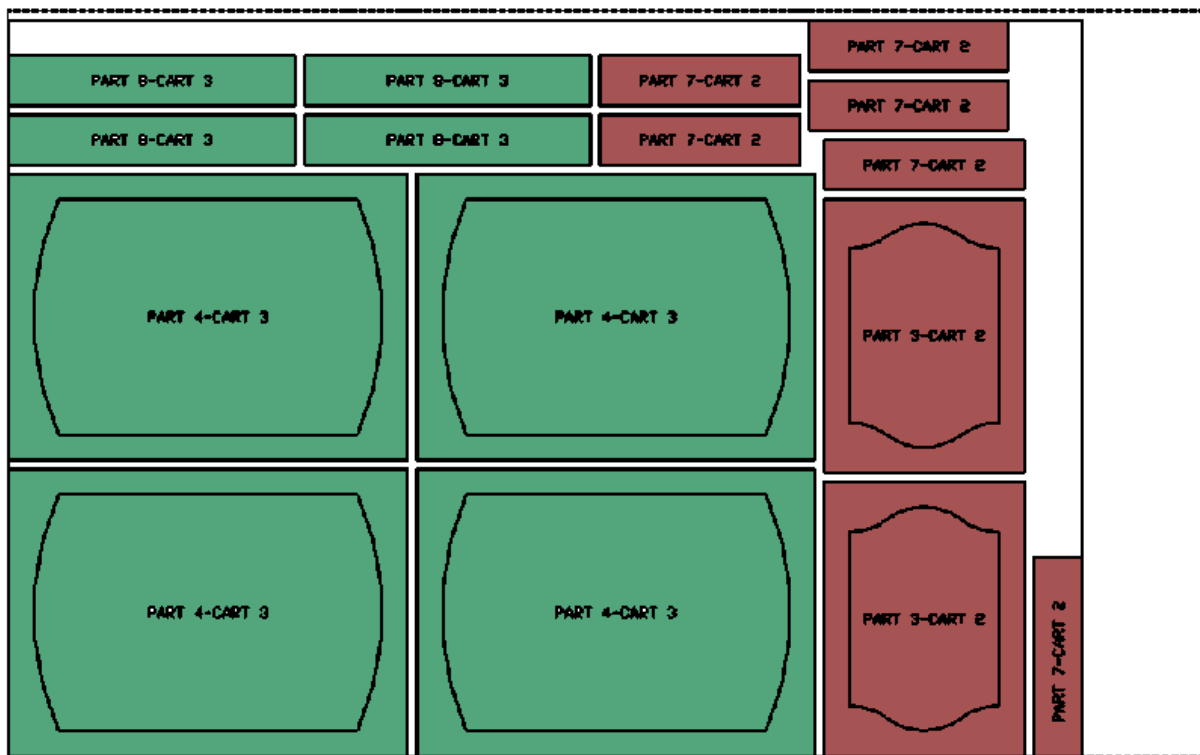
This example will use a job that has 10 parts that belong to 5 different carts. The breakdown is like this:

	Part#	Part	Qty	Material		X Dim	Y Dim	Z Dim	Full Path to Part
	1	STYLE0.SCN	12	3/4 MDF	CART 1	12	4	.75	C:\rcim_work\STYLE0.SCN
	2	Style1SQ.scn	12	3/4 MDF	CART 1	12	16	.75	C:\rcim_work\Style1SQ.scn
	3	STYLE2.SCN	8	3/4 MDF	CART 2	16	22	.75	C:\rcim_work\STYLE2.SCN
	4	STYLE6.SCN	4	3/4 MDF	CART 3	23	32	.75	C:\rcim_work\STYLE6.SCN
	5	STYLE8.SCN	12	3/4 MDF	CART 4	19	23	.75	C:\rcim_work\STYLE8.SCN
	6	STYLE9.SCN	10	3/4 MDF	CART 5	18	24	.75	C:\rcim_work\STYLE9.SCN
	7	STYLE0.SCN	8	3/4 MDF	CART 2	16	4	.75	C:\rcim_work\STYLE0.SCN
	8	STYLE0.SCN	4	3/4 MDF	CART 3	23	4	.75	C:\rcim_work\STYLE0.SCN
	9	STYLE0.SCN	12	3/4 MDF	CART 4	19	4	.75	C:\rcim_work\STYLE0.SCN
	10	STYLE0.SCN	10	3/4 MDF	CART 5	18	4	.75	C:\rcim_work\STYLE0.SCN

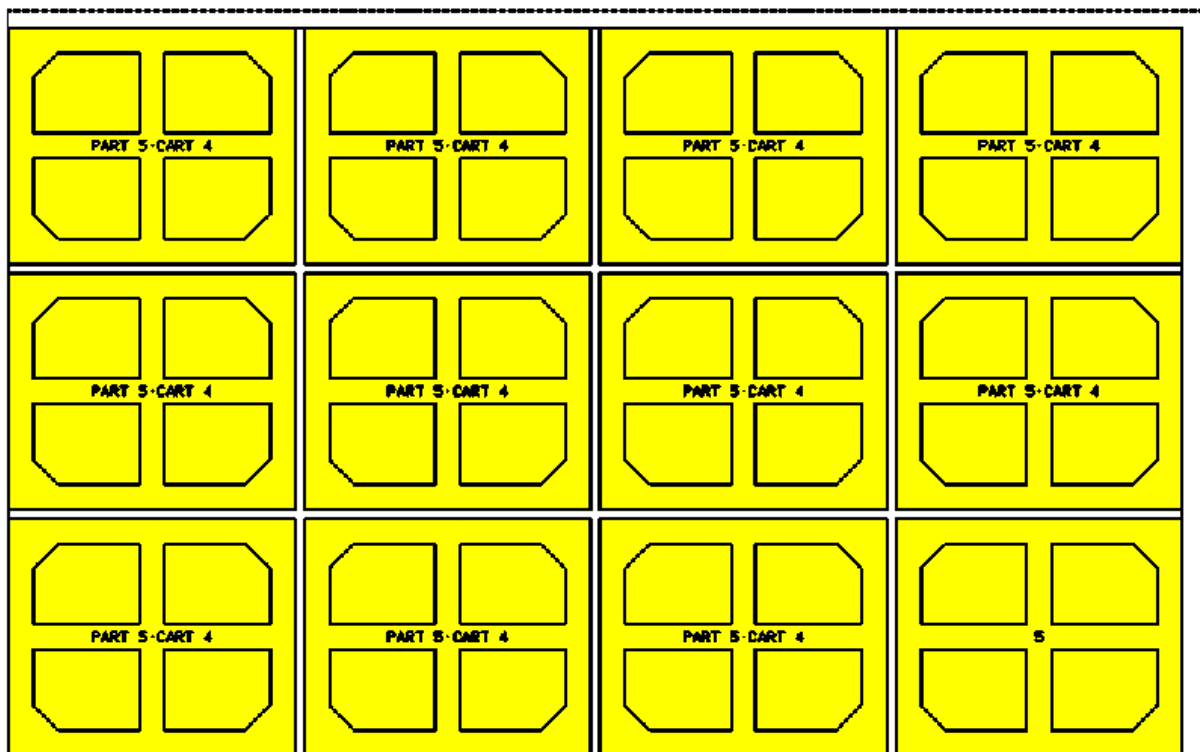
Once this job is nested with the previously set parameters the sheets will look like this:

PART 7-CART 2	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1
PART 7-CART 2	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1	PART 1-CART 1
PART 3-CART 2	PART 3-CART 2	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1
PART 3-CART 2	PART 3-CART 2	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1
PART 3-CART 2	PART 3-CART 2	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1	PART 2-CART 1

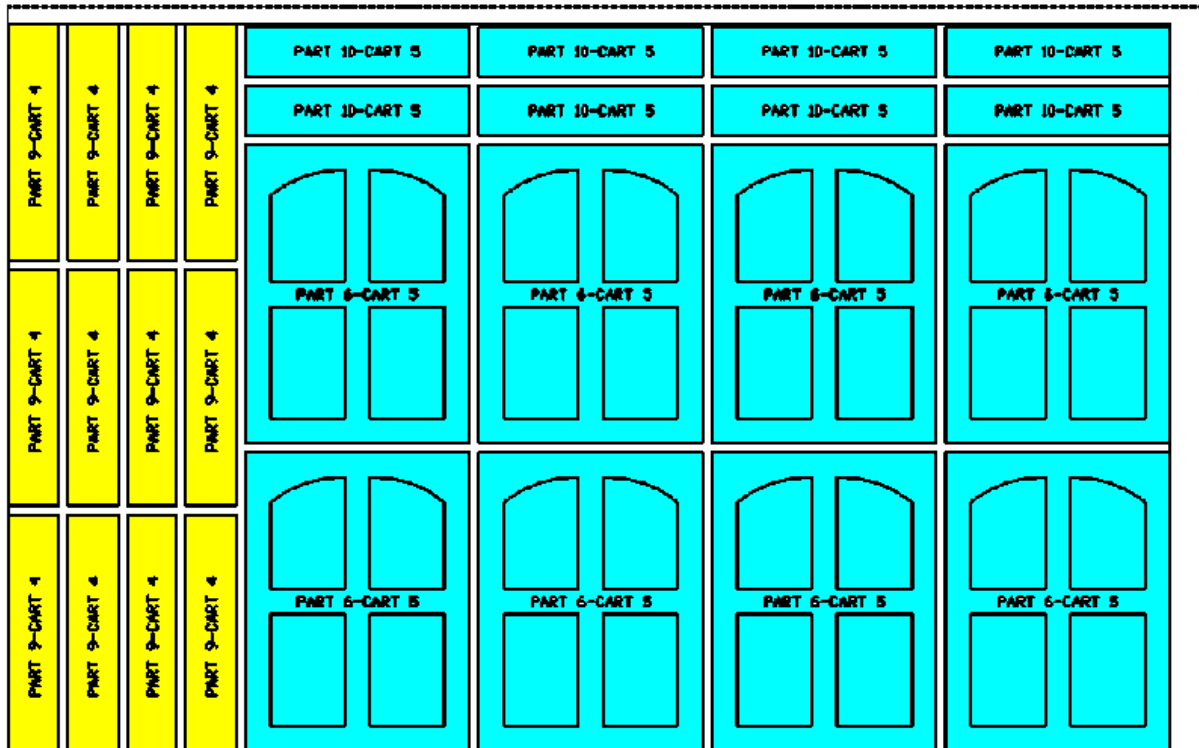
Sheet 1 has parts 1, 2, 3, and 7 which are from Cart 1 and Cart 2. Cart 1 is finished.



Sheet 2 has parts 3, 4, 7 and 8 which are from Cart 2 and Cart 3. Cart 2 is finished. Cart 3 is also finished.



Sheet 3 has part 5 which is from Cart 4 only.

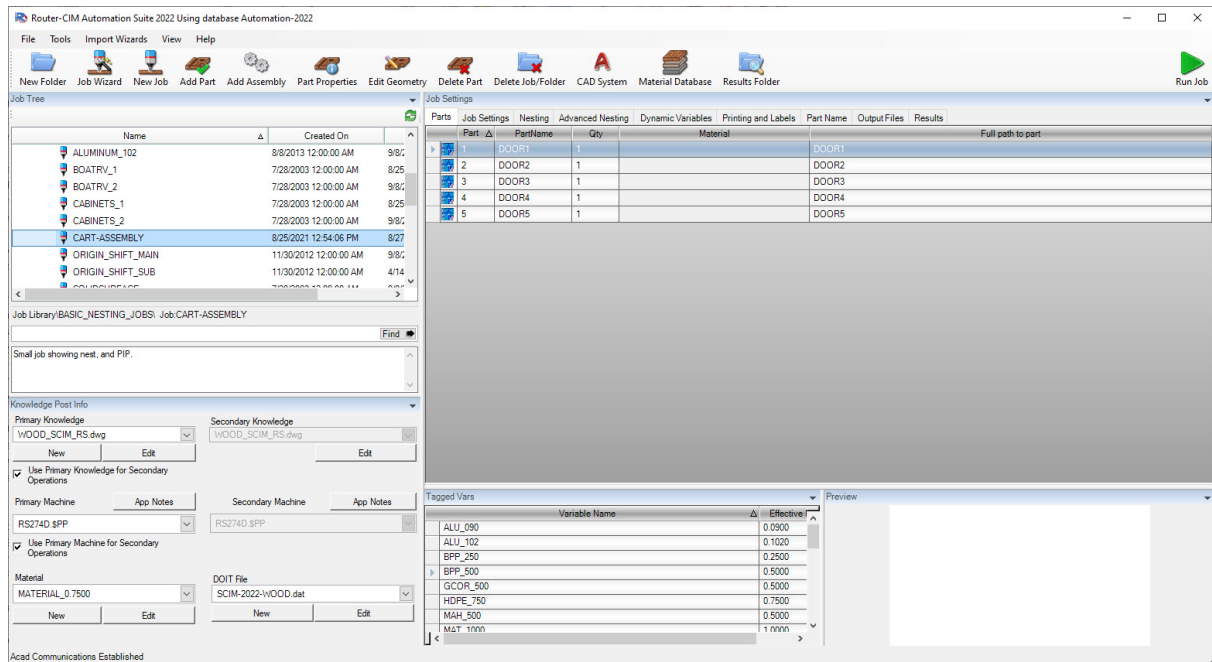


Sheet 4 has parts 6, 9, and 10 which are from Cart 4 and Cart 5. Cart 4 is finished.

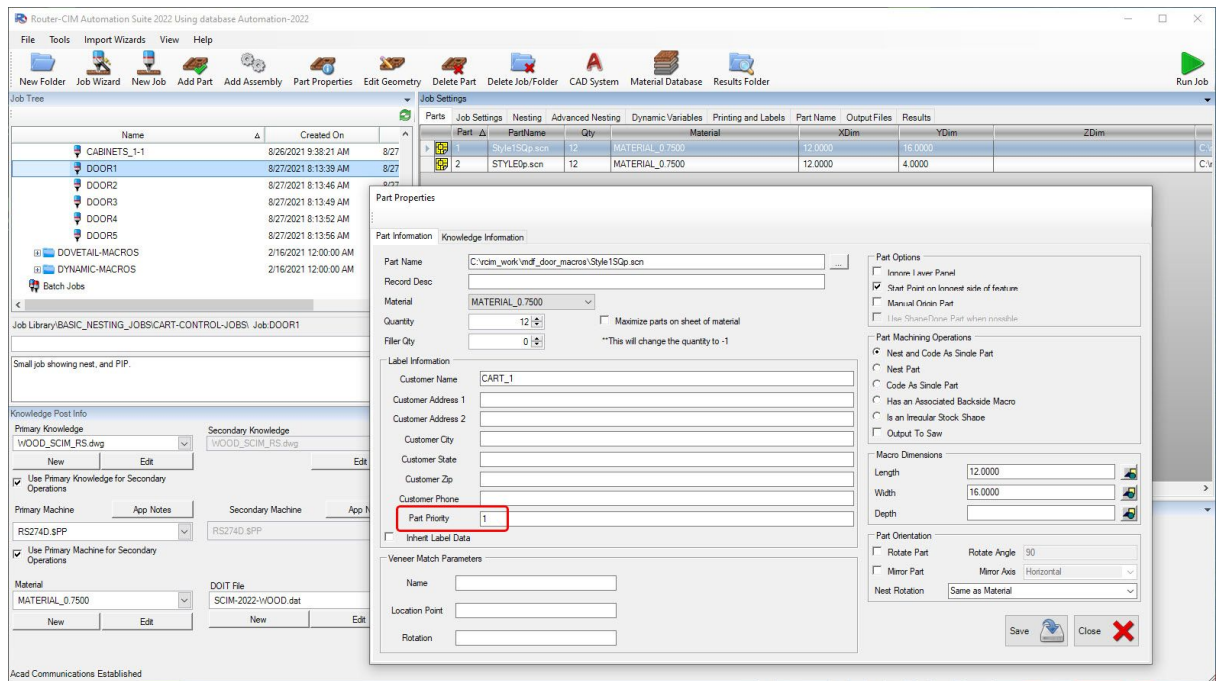


Cart Control can also be used in an assembly job. Instead of adding all your parts into a single job, you can add different assemblies to a job. This creates a master job. Cart Control follows the same part priority structure as a standard job. You will need to define the Part Priority as mentioned above on a part level basis. The parts in the assembly jobs will be merged into a large job and the priorities will then be defined. If all the assemblies added to the master job have a part priority that matches, those parts will be nested together.

Here is a master job that includes assemblies to be used with Cart Control.

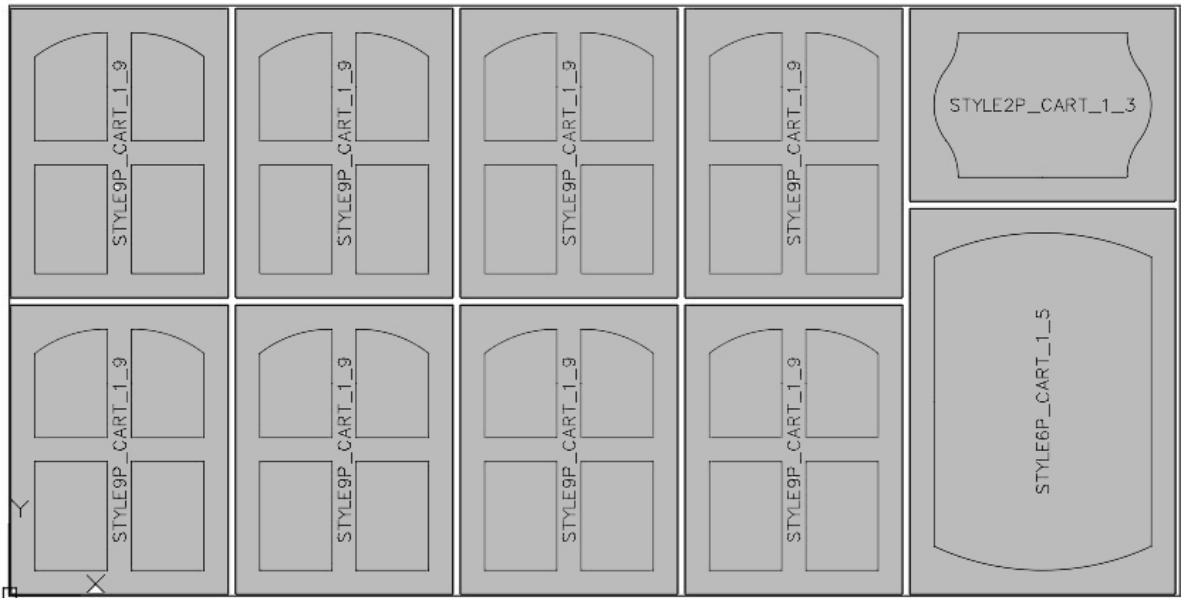


In each assembly that was added, each part has been cart.

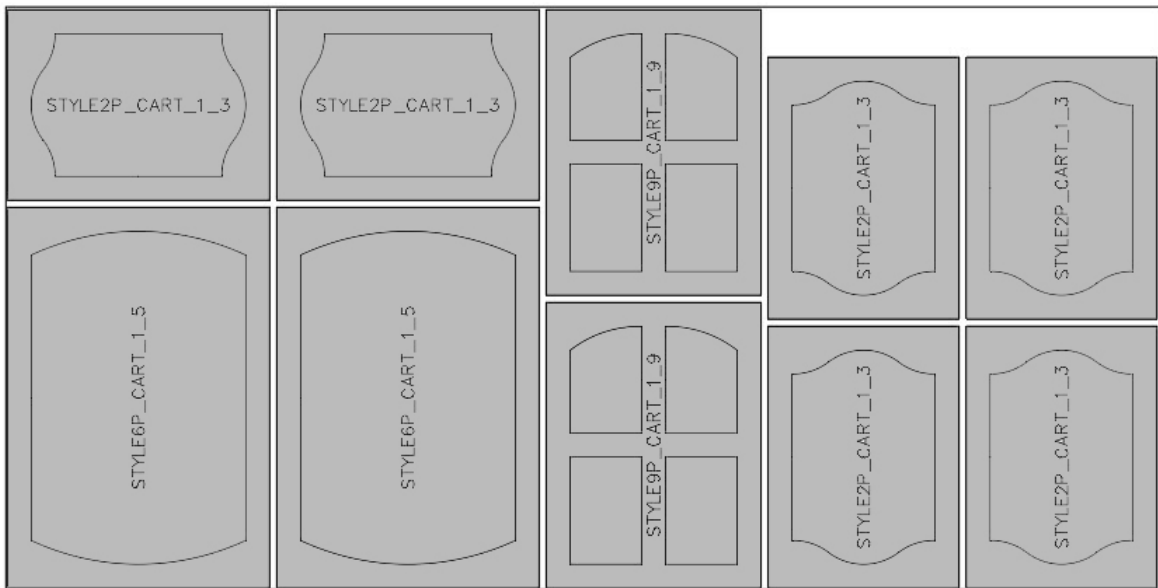


The resulting cart control nest is as follows:

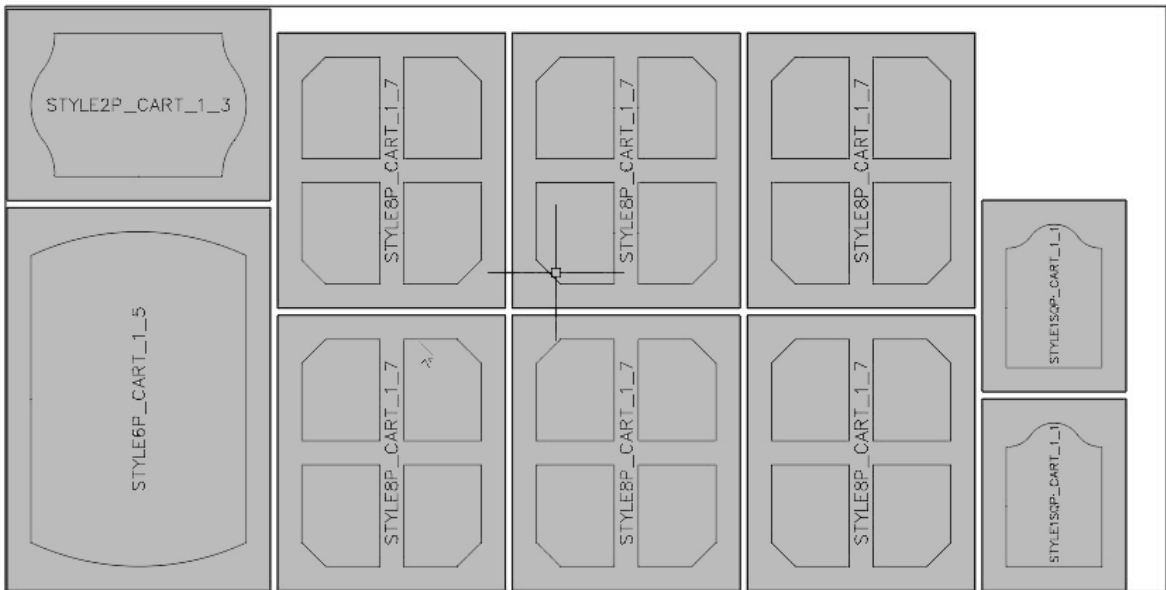
Sheet 1 has Cart 1 parts:



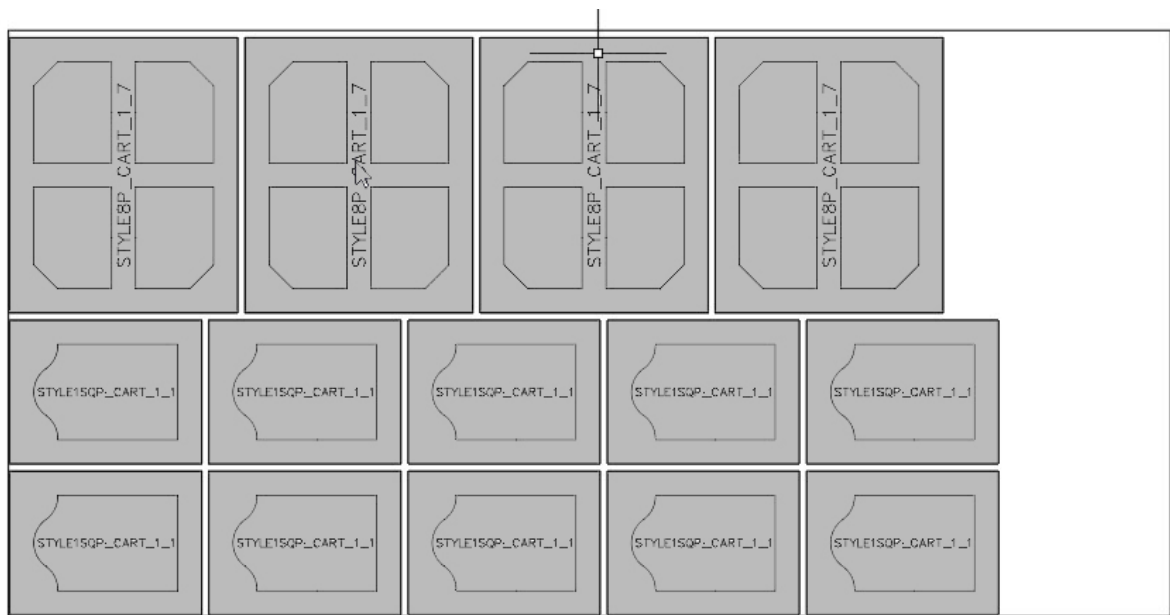
Sheet 2 has Cart 1 parts:



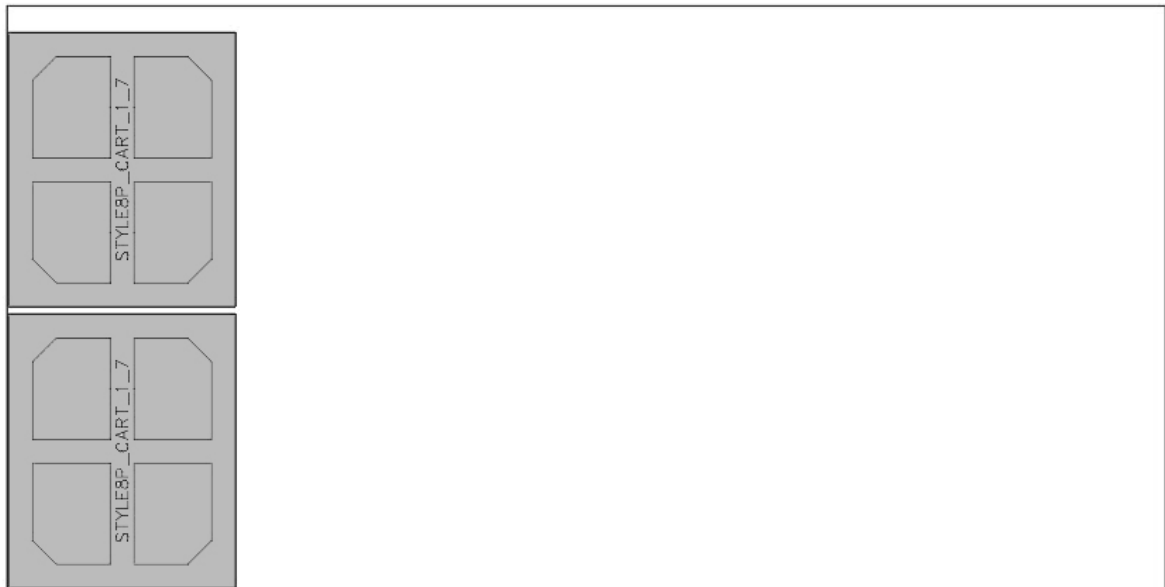
Sheet 3 has Cart 1 parts:



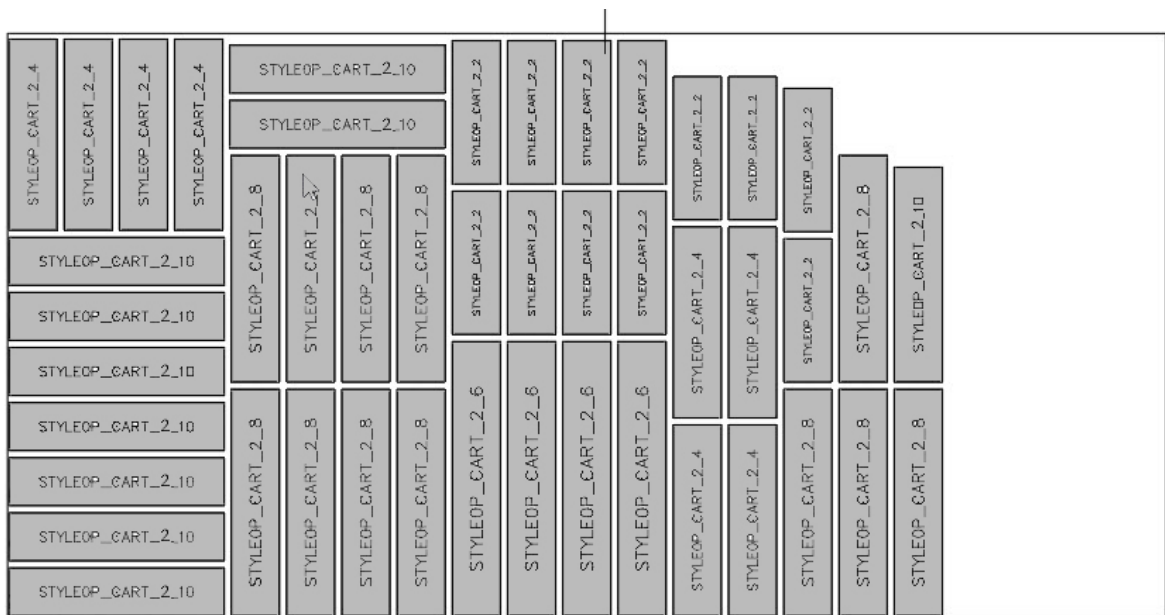
Sheet 4 has Cart 1 parts:



Sheet 5 has Cart 1 parts:



Sheet 6 has Cart 2 parts:



Veneer Matching Parameters (Grain Matching)

A veneer matched set/group is the combination of multiple parts that you want to be nested together in a set configuration so that they are machined from one section of a sheet.

Veneer Match Parameters

☒ Display Individual Parts
☐ Display Outline of Veneer Match
☐ Display Parts and Outline of Veneer Match

Veneer Match Bridge

The result of the Veneer Match Parameters are shown here:

Display Individual Parts

Parts are nested individually using the materials bridge width to separate the parts:



Display Outline of Veneer Match

Parts are not shown and an outline of the veneer match set is shown:

The layer used to define the outline of the veneer match is VENEER_OUTLINE.

Note: If '[Label Parts](#)' is selected under the Nesting tab, the veneer match outline will be identified by the veneer match name per set.



Display Parts and Outline of Veneer Match

Parts are nested individually using the materials bridge width to separate the parts and an outline of the veneer match set is shown:

The layer used to define the outline of the veneer match is VENEER_OUTLINE.

Note: If '[Label Parts](#)' is selected under the Nesting tab, the veneer match outline will be identified by the veneer match name per set.



Veneer Match Bridge

To override the Veneer Matching default of using the material's bridge width for part spacing. This should be a real number such as 0.5 or 0.125 for example. If the value is 0.0 (default) the veneer match bridge will use the material's bridge width setting.

To use the Veneer Match Location points exactly as they have been defined, you will add an NCVAR called *NOEXPAND*. When this variable is added, the Veneer Match Location Points will be honored exactly with no spacing based on the material's bridge width or Veneer Match Bridge setting. For information on adding the *NOEXPAND* variable, refer to the ['Common NCVARs in Automation'](#) section.

The information that is needed in order to correctly let nesting know what parts need to stay together can be accomplished in two ways.

- 1) Manually find the Identification (ID) points and fill in the correct Part Properties parameters for the Veneer Match function.
- 2) Use Solid-CIM 3D on a 3D Model/Assembly so it can define the Identification (ID) points for you so they can be imported to the right Part Property parameters. Refer to the Solid-CIM 3D Help Manual for more information on how to use the Veneer Matching feature through Solid-CIM 3D.

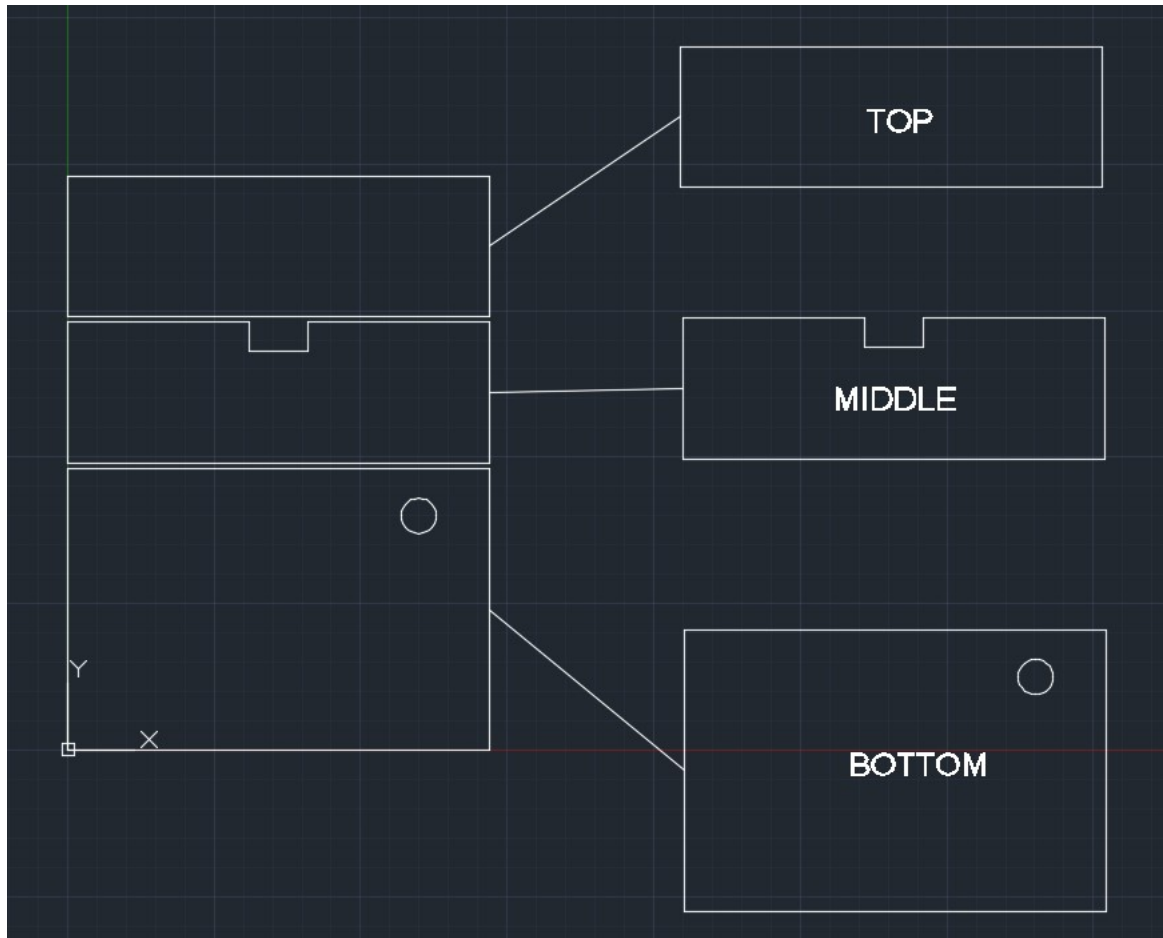
Veneer Matching Considerations:

- 1) You cannot use ShapeDone with parts that will be veneer matched. If ShapeDone is turned on for those specific parts it will still be processed as if it was not turned on.
- 2) The spacing between each part that is veneer matched will match the material bridge width. If a different spacing is desired you can set the Veneer Match Bridge to the desired spacing for the veneer matched parts.
- 3) Veneer match rotation is defaulted to 0, if a different rotation is desired reference Figure 6 for how rotation works with veneer matched parts.

Manually Creating Veneer Matching Parts:

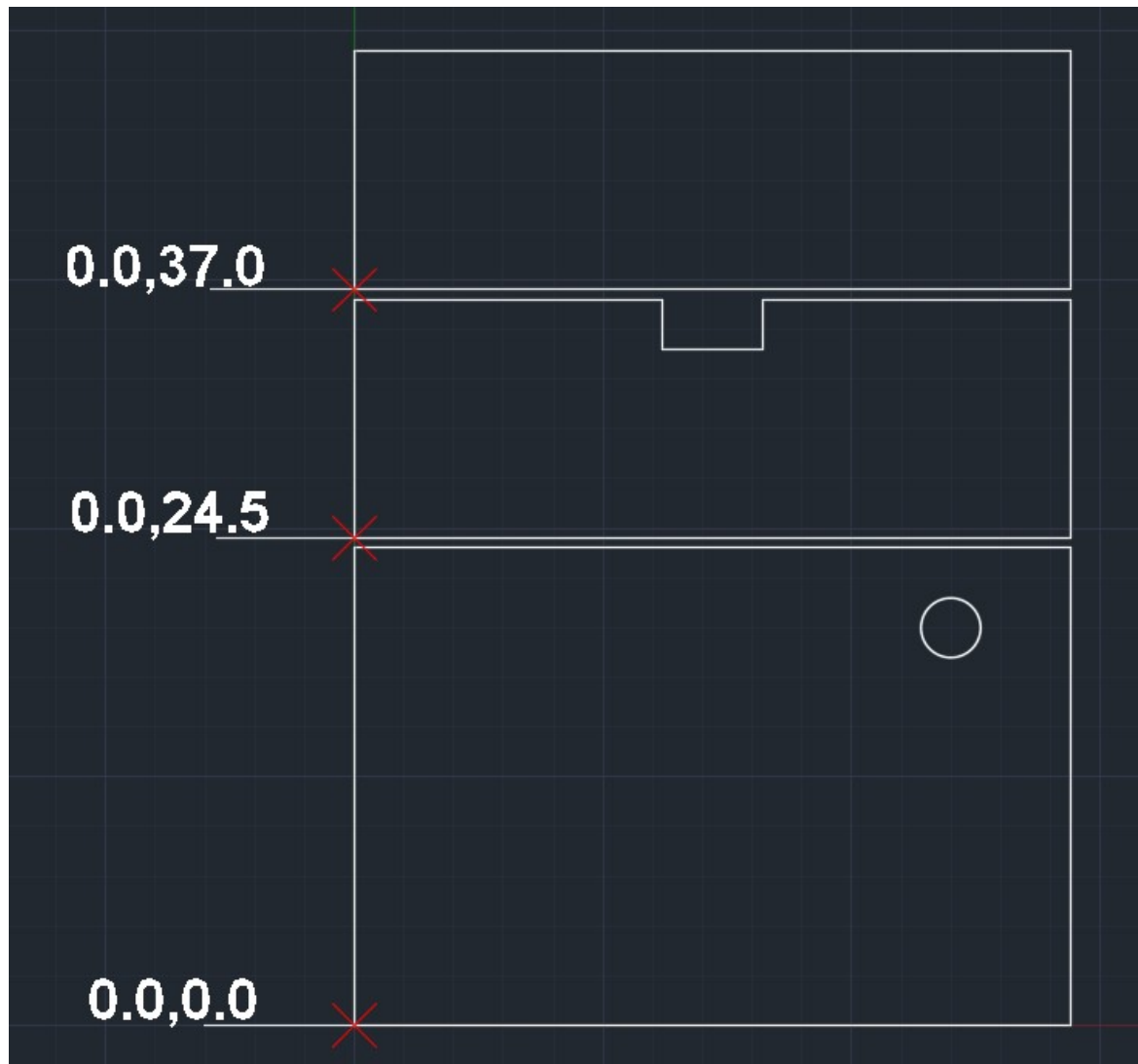
- 1) Create your individual parts that will be included in the final veneer matched part configuration. Each part should be in its own drawing for Router-CIM Automation Suite.
- 2) Once all of the parts are created, insert those individual parts into a new drawing in the configuration you would like nested. This is shown below, where we are using the three parts on the right: Top, Middle, and Bottom to make the configuration on the left.

Note: The drawing you are inserting the parts to be Veneer Matched will not need to be saved. This drawing is only used to obtain the information needed for the Veneer Matching Parameters.



- 3) Once the parts are configured the way that you want them, like shown on the left of the above image, use the ID command you select the lower left point of each part, and record the X, Y, Z value. We are only looking for 2D points, so make sure you ignore whatever value represents the thickness of the part. In the example shown we are ignoring the Z-Value because the drawing is in the XY plane with thickness being the Z dimension. Below are the results for this command on the lower left point of each part.

Note: Make sure the parts are configured in the AutoCAD drawing as if they were laid out on the sheet of material you will be using. If the veneer/grain direction on the sheet of material follows the long side of the sheet, make sure the parts are configured to match.



- 4) Once you have the location point information for each part you should now bring these parts into Router-CIM Automation Suite.
- 5) Fill in the veneer match parameters for all parts that should be veneer matched for each individual part. These settings are located under the Part Properties of each part.

There are three Veneer Match Parameters available:

A) **Name (Required)** - This parameter is used to define which parts need to be Veneer Matched when going to the nesting function. In this instance, all three parts are defined as MATCH1. The Veneer Match feature can handle multiple sets/groups in a single job. Each set/group would need its own unique name in order for them to be nested correctly.

B) **Location Point (Required)** - This is the relative location of each part in regards to every other part in the veneer matched set/group. The appropriate format of this field is Coordinate 1 (X

Axis) followed by a comma and then Coordinate 2 (Y Axis) (i.e. 3.0,4.0). No spaces are allowed. This information was obtained in step 3 of this section.

C) Rotation (If needed) - The first part in the veneer match set/group with a Rotation will be used to direct Router-CIM Automation Suite on how you want the veneer matched part rotated. All parts in the veneer match part should have the same number to avoid errors when processing. This is only needed if the parts in the veneer match set/group were not configured correctly to match the direction of the veneer/grain direction of the material being used.

Part 1 (BOTTOM):

Veneer Match Parameters

Name

Location Point

Rotation

Part 2 (MIDDLE):

Veneer Match Parameters

Name

Location Point

Rotation

Part 3 (TOP):

Veneer Match Parameters

Name

Location Point

Rotation

- 6) The final step is to configure your material to allow for Veneer Matching. Simply go into your material(s) that you will be using and under the Advanced Nesting Parameters for the material, check the box for 'Allow Veneer Matching'.

Scrap Management Advanced Nesting Parameters

Skeleton-Cut-Off Parameters

Enable Skeleton-Cut-Off ☐

Offset Distance

X Spacing

Y Spacing

Equal Spacing ☐

of X Spacings

of Y Spacings

Multi-Stock Nesting

☒ Priority ☐ Best Yield ☐ Best Price

☐ Automatically Decrease Qty

Alternate Sizes

Add Alternate Size Delete Alternate Size

YDim	XDim	Qty	Priority	Cost
49.0000	121.0000	999	8	0.0000
61.0000	97.0000	999	10	0.0000
61.0000	121.0000	999	10	0.0000

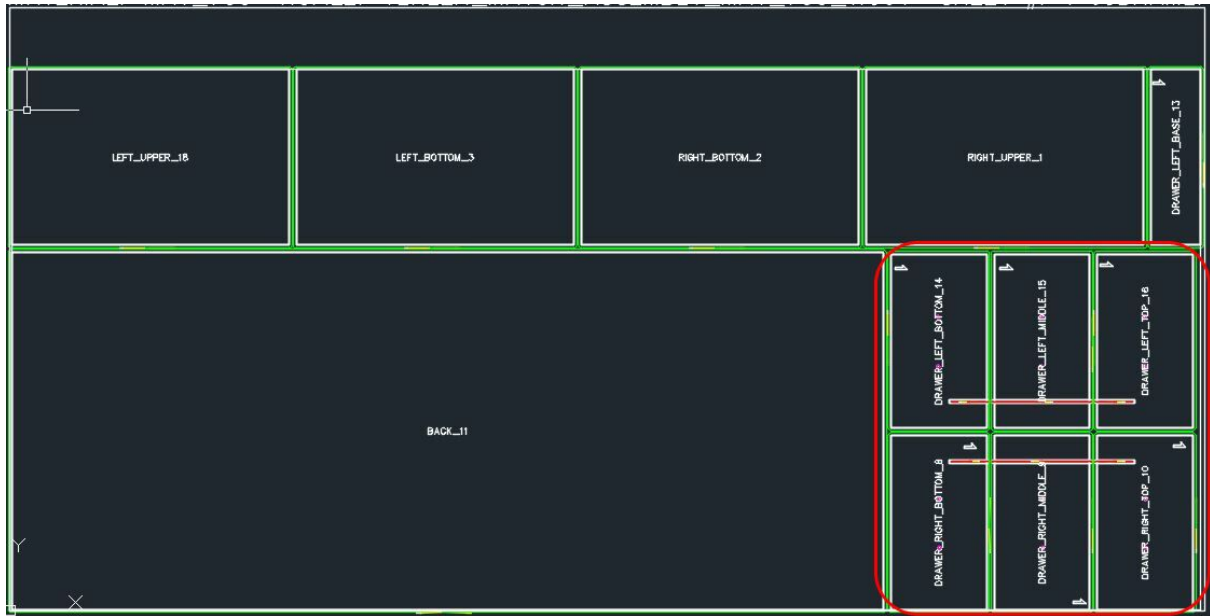
Cart Control Parameters

Jobs per Cart

Open Cart Threshold

☒ Allow Veneer Matching

The result of the Veneer Matching process is shown here:



Small Part Zones

Small Part Control Nesting allows a user to define locations where they want small parts to be nested.

Note: In order to use the Small Part Zones option, you will need ['Rename Outside Layers'](#) defined. Refer to the ['Job Settings'](#) section for more information.

Small Part Zones

☒ Entire Part Inside
 ☐ Center of Gravity Inside

Zone Extension Allowed (%)

Small Part Edge Offset

Part Area Definition

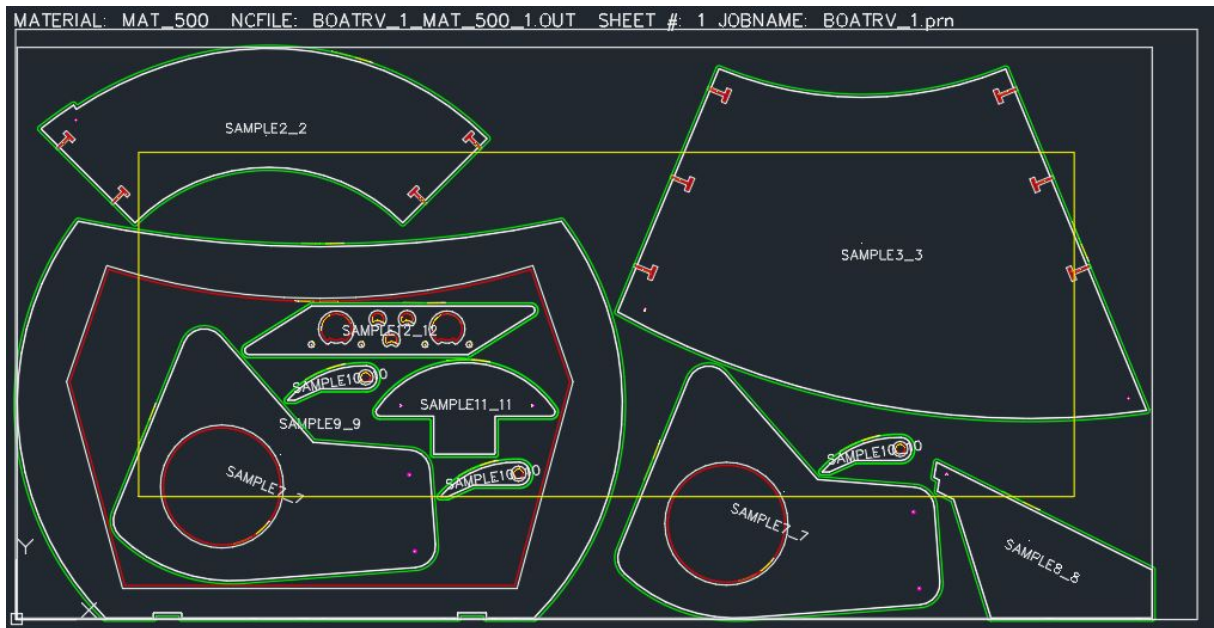
When a part is identified as small by the Rename Outside Layers option, the Small Part Zone feature place the small part within the boundary based on either the entire outside geometry of the part (Entire Part Inside) or on the parts identified center of gravity (Center of Gravity Inside). Default is 'Entire Part Inside'

Zone Extension Allowed (%)

Allows the user to apply a zone extension as a percent to the Small Part Zone. If the Zone Extension Allowed is defined as 10.0, it will allow a 10% extension around the defined Small Part Zone. For example, if the small part zone has a dimension of 10 by 10, the zone extension would allow nesting to look at an 11 by 11 zone.

Small Part Edge Offset

Defining a Small Part Edge Offset will use the material size and offset it by the number input here. Use this option to create a boundary based on the material size.



Create a Custom Small Part Zone

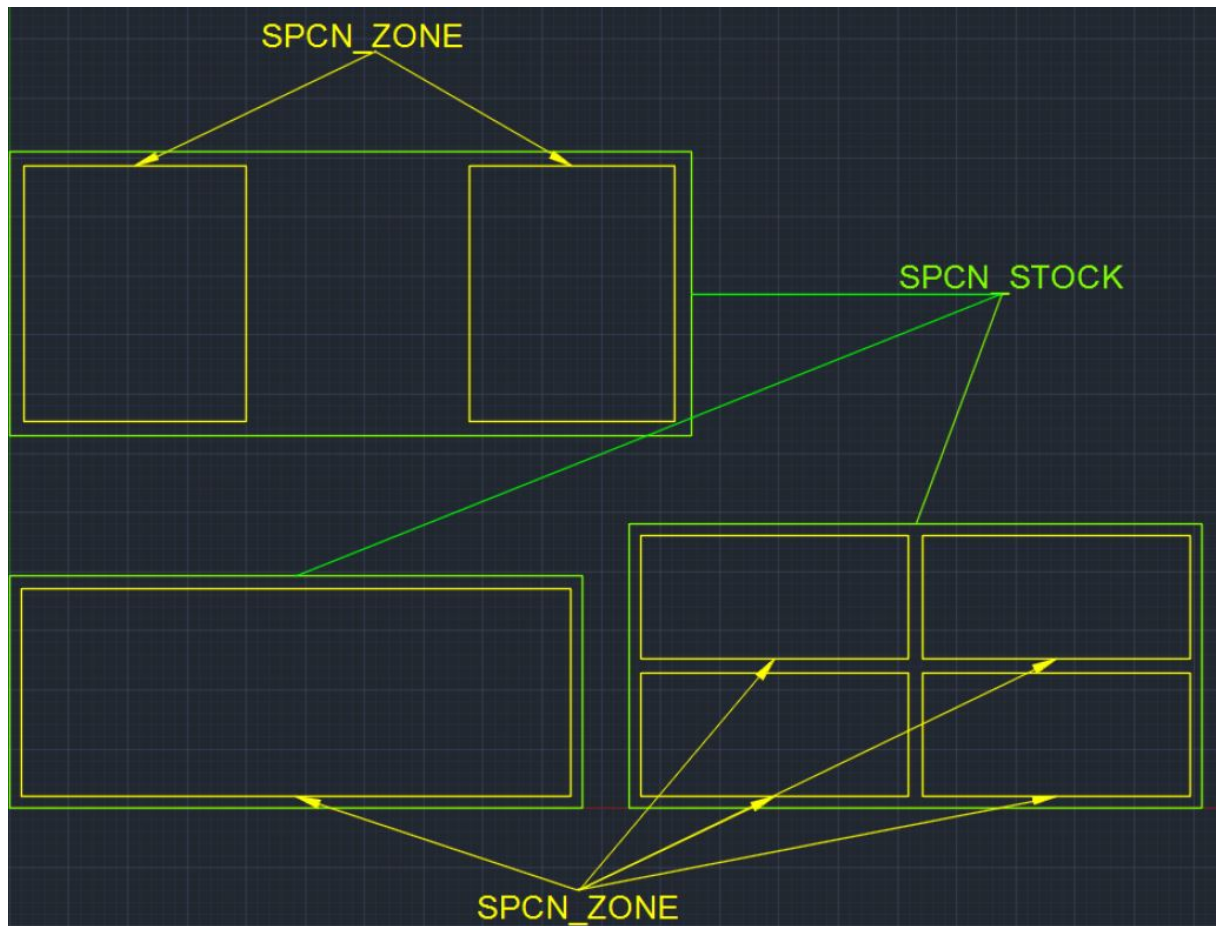
Custom Small Part Zones can be defined by creating an AutoCAD drawing and following these steps.

Create an AutoCAD Drawing

Create an AutoCAD drawing that defines the stock size and the small part zones.

When creating this drawing, the stock size needs to match the size of the material that it is being defined for. When the stock size is drawn, it will need to be located on a layer called SPCN_STOCK to identify the overall size of the material.

Next you will draw the small part zones with the geometry on the layer SPCN_ZONE. You can identify up to 4 unique zones.



Note: The stock/material and the small part zones need to be closed geometry rectangles.

You will then need to save the drawing with a specific name. The name of the drawing will **MUST** to be the MATERIAL CODE as defined in the Material Database (See below).

Once the drawing has been saved, you will need to add the drawing to the Router-CIM Automation Suite job as a part. For more information on how to add a part to a job, go to the ['Add Part'](#) section under ['Toolbar'](#).

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
	Part #	PartName	Qty	Material				
	1	Sample1.dwg	4	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample1.dwg		
	2	Sample2.dwg	1	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample2.dwg		
	3	Sample3.dwg	2	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample3.dwg		
	4	Sample4.dwg	2	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample4.dwg		
	5	Sample5.dwg	4	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample5.dwg		
	6	Sample6.dwg	1	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample6.dwg		
	7	Sample7.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample7.dwg		
	8	Sample8.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample8.dwg		
	9	Sample9.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample9.dwg		
	10	Sample10.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample10.dwg		
	11	Sample11.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample11.dwg		
	12	Sample12.dwg	3	MATERIAL_0.5000		C:\rcim_work\Automation\Samples\DWGS\Sample12.dwg		
	13	MAT_500.dwg	1	MATERIAL_0.5000		C:\rcim_work\SPCN\MAT_500.dwg		

Material Properties	
Material Description	MATERIAL_0.5000
Material Code	MAT_500
Material Handling Code	
Stock Settings	
Sheet Stock Y Dim	48.0000
Quantity	999
Sheet Stock X Dim	96.0000
Priority	5
Thickness	0.5000
Bridge Width	0.6250
Cost	0.00

With the drawing added to the job AND placed on the correct material, custom Small Part Zones will be used. An example is shown here.



STAYDOWN and PIP (Part Inside Part) Bridging

When using STAYDOWN and PIP (Part Inside Part) nesting, the STAYDOWN path can be applied to parts that have been nested within other parts. This setting will change how the STAYDOWN path is applied to these parts.

Note: This setting is only for when parts are nested within other parts (PIP nesting). All normal STAYDOWN paths will be created for parts that are not nested within other parts.

Bridge Parts to One Another and PIP Layer

Staydown and Part Inside Part Bridging

- ☒ Bridge Parts to One Another and PIP Layer
- ☐ Bridge Parts to One Another Only
- ☐ Disable Bridging For Part Inside Part

This setting will create a STAYDOWN path that will connect all the parts within the PIP layer AND the PIP layer itself.

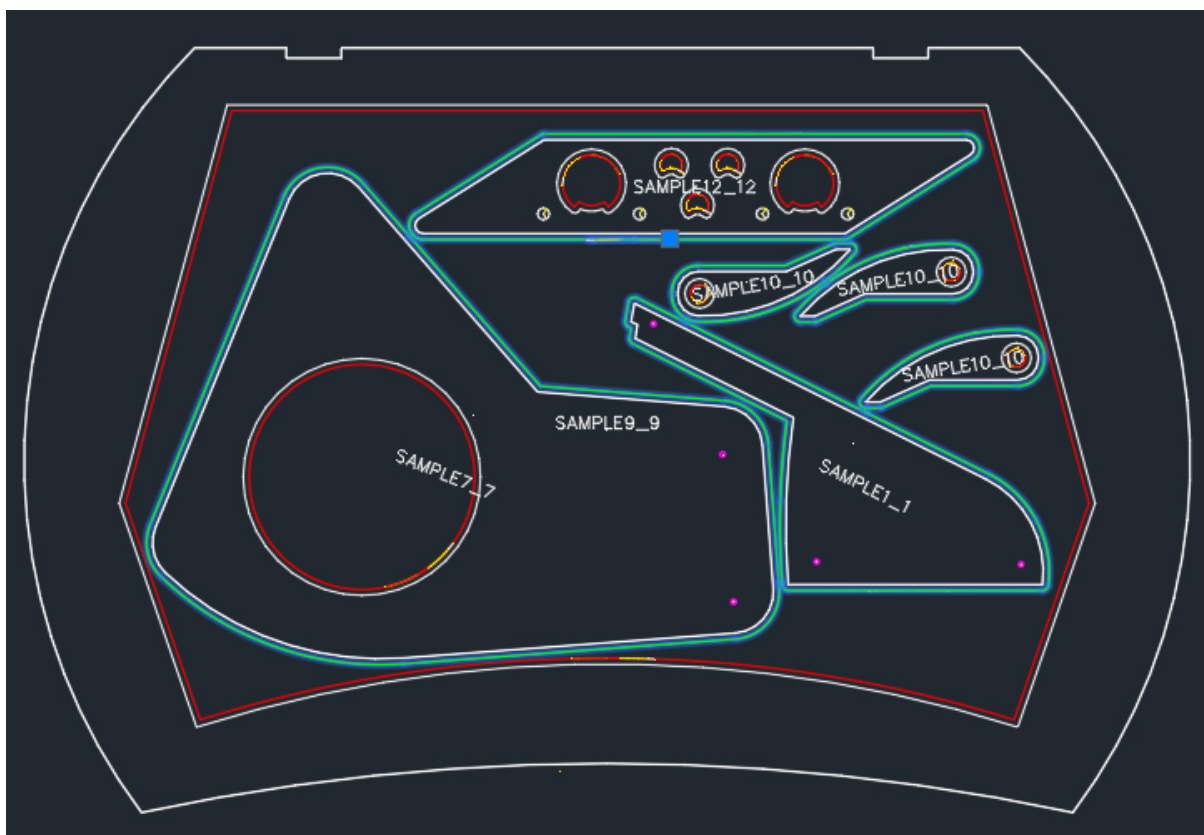


Bridge Parts to One Another Only

Staydown and Part Inside Part Bridging

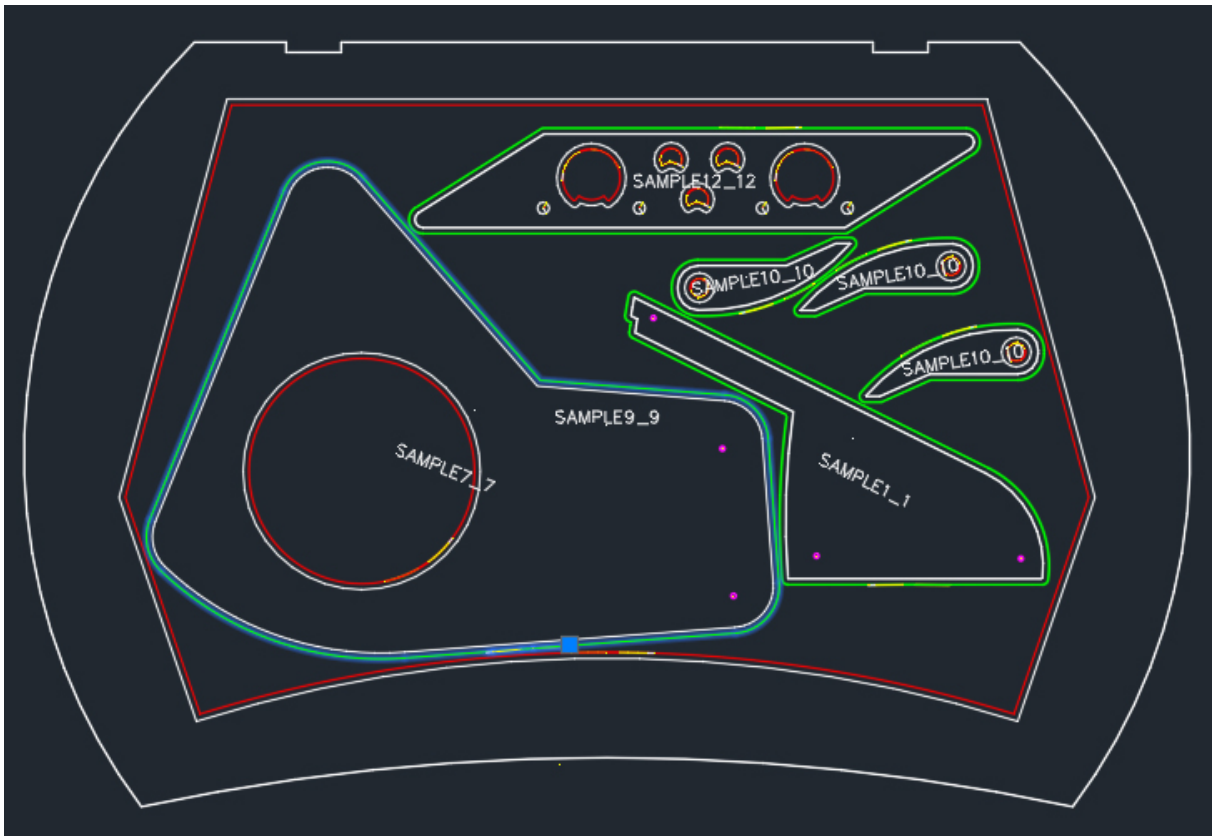
- ☐ Bridge Parts to One Another and PIP Layer
- ☒ Bridge Parts to One Another Only
- ☐ Disable Bridging For Part Inside Part

This setting will create a STAYDOWN path that will connect ONLY the parts within the PIP layer.

**Disable Bridging For Part Inside Part****Staydown and Part Inside Part Bridging**

- ☐ Bridge Parts to One Another and PIP Layer
- ☐ Bridge Parts to One Another Only
- ☒ Disable Bridging For Part Inside Part

This setting will NOT create a STAYDOWN path when parts are nested within other parts.



Skeleton Cutting Controls

The Skeleton Scrap Cutting feature will allow the user to define a maximum size of scrap that they will want the Router-CIM Automation Suite to leave within the nest.

When a skeleton cut is defined, you can skeleton cut the nest based on the entire sheet size or the enclosing rectangle which is the amount of area used by the nested parts themselves.

For more information on how to set up the Skeleton Scrap Cutting feature, refer to the ['Advanced Nesting Material'](#) section.

Whether you select the 'Entire Sheet' or 'Enclosing Rectangle' option, you will need to define a Minimum X and Minimum Y.

Minimum X - Identifies the minimum amount from the edge of the enclosing rectangle to the edge of the sheet for Skeleton to extend to the edge in the X direction

Minimum Y - Identifies the minimum amount from the edge of the enclosing rectangle to the edge of the sheet for Skeleton to extend to the edge in the Y direction

If the value found is larger than the minimum identified, skeleton cutting will not extend to the edge of the sheet.

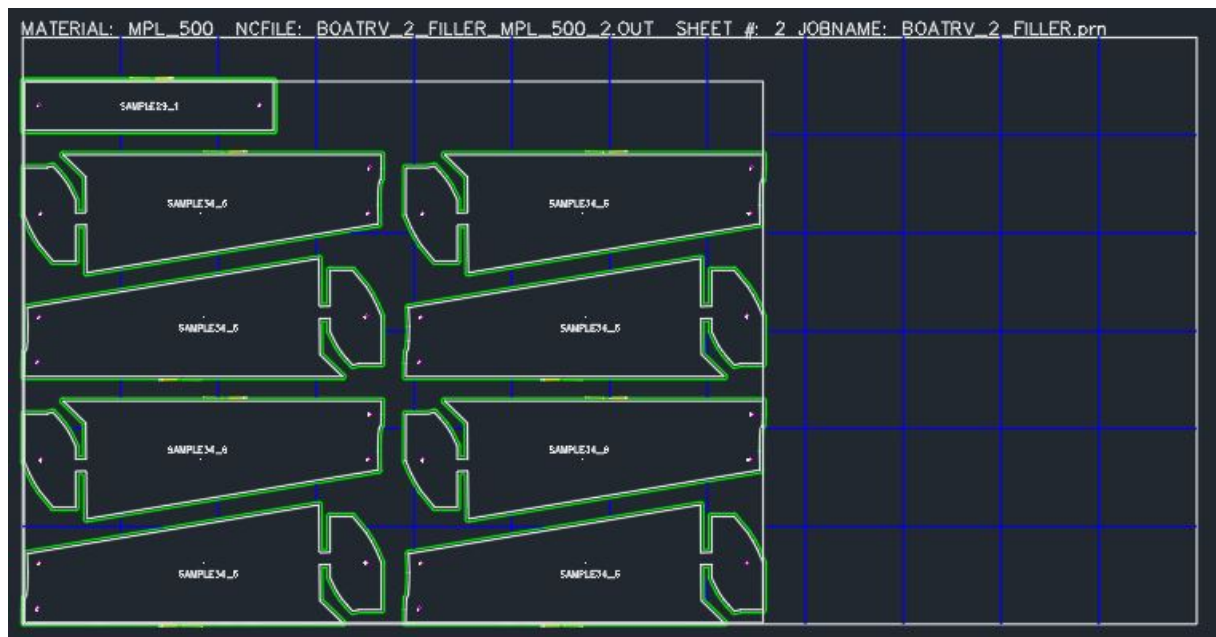
Entire Sheet:

Skeleton Cutting

☒ Entire Sheet ☐ Enclosing Rectangle

Minimum X

Minimum Y



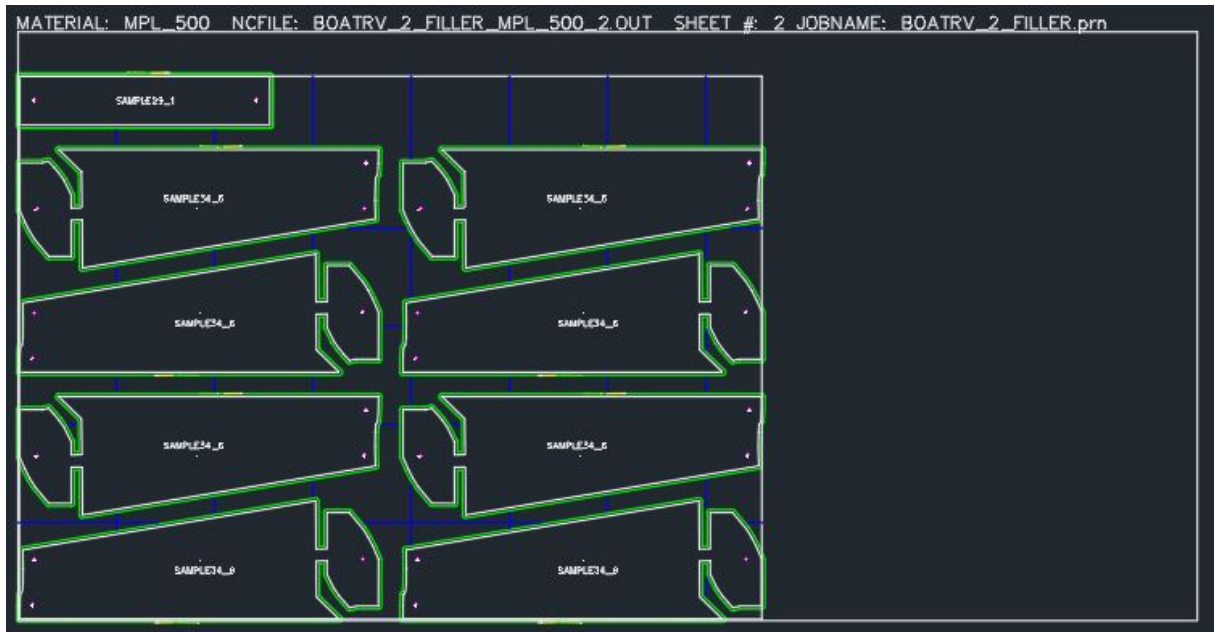
Enclosing Rectangle:

Skeleton Cutting

☐ Entire Sheet ☒ Enclosing Rectangle

Minimum X

Minimum Y



Dynamic Variables

	Name	Value
▶	MDF75	.75
	MEL750	.75
	PLY250	.25
	MDFXX	.75
	WHT75	0.750
	BIR25	0.25
	MEL75	0.750
	ALM75	0.750
	BBP25	0.2187
	MEL25	0.25

This section will show all the Dynamic Variables from a macro job or part. If there are no macros in the job, the default will show each of the materials as the materials can be used for setting thickness in a macro.

Printing and Labeling

The Printing and Labeling settings allow you to adjust the Label Output, Printing options and Reporting features.

Parts Job Settings Nesting Advanced Nesting Dynamic Variables **Printing and Labels** Part Name Output Files Results

Labeling

Labels

Customer Info 1 JobLabel1

Customer Info 2 JobLabel2

Customer Info 3 JobLabel3

Customer Info 4 JobLabel4

Customer Info 5

Customer Info 6

Customer Info 7

Customer Info 8

RCIM Labeling

Active Label Configurations		
Active	Name	OutputTNP
<input checked="" type="checkbox"/>	RCIM_DefaultLabeling	<input type="checkbox"/>
<input type="checkbox"/>	RCIMLabel	<input type="checkbox"/>

Assembly Labels

☒ Use Top Down Label Data Precedence

☐ Use Bottom Up Label Data Precedence

☒ Print Sheet Quantity on Label

☒ Print One Barcode Label per Sheet

Printing

Select Printer Brother MFC-6490CW Printer

☒ Landscape ☐ Portrait ☐ Auto

☐ Print Nests ☒ Create Nest PDF

☐ Print Code As Single Parts ☐ Create Code as Single Part PDF

☐ Print Material Usage Report ☐ Create Material Usage Report PDF

Header Text Size 1.0000

Combine PDFs

☐ Combine All PDF ☐ Combine Code as Single PDF

☐ Combine Nest PDF ☐ Combine Nest PDF by Material

Reporting

☐ Reset Sheet Counter for Each Material

☒ Create Nest Summary Report

☐ Create Single Part Summary Report

Labels

This section is where you can place any custom data you wish to appear on your labels for each part if you are using **'Inherit Label Data from Job'** under Part Properties.

Labeling

Labels

Customer Info 1 JobLabel1

Customer Info 2 JobLabel2

Customer Info 3 JobLabel3

Customer Info 4 JobLabel4

Customer Info 5

Customer Info 6

Customer Info 7

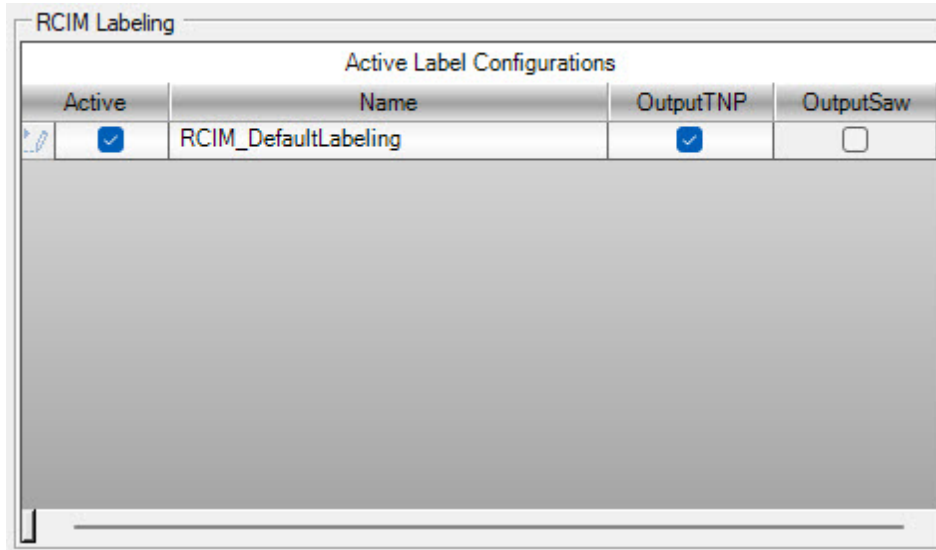
Customer Info 8

RCIM Labeling

This section is where you select the RCIM Label Configuration when using the ['Label Designer'](#).

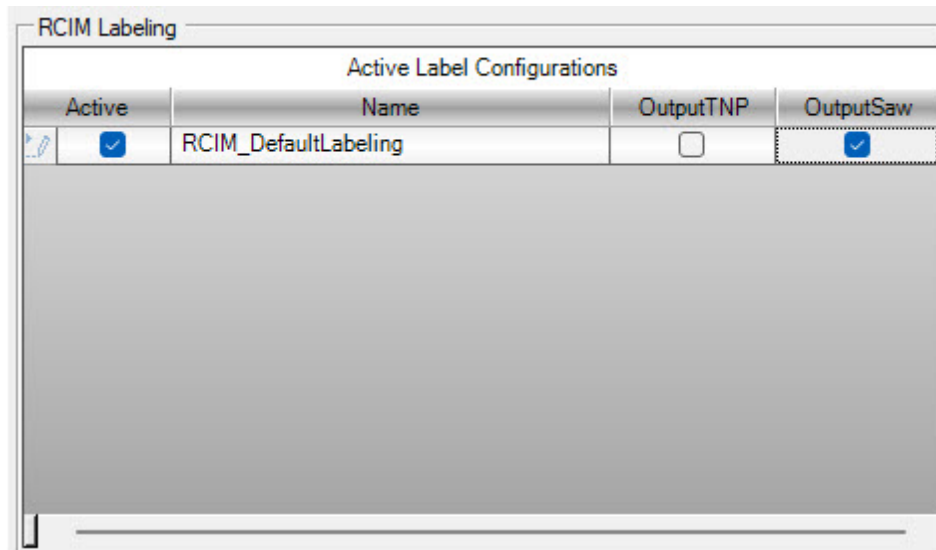
Check the RCIM Label Configuration that you want active for this job.

OutputTNP: If you have the CIM-Tech Touch-N-Print system, you can also select if you want to use one of your custom label setups for this option by checking the 'OutputTNP' box.



Active Label Configurations			
Active	Name	OutputTNP	OutputSaw
<input checked="" type="checkbox"/>	RCIM_DefaultLabeling	<input checked="" type="checkbox"/>	<input type="checkbox"/>

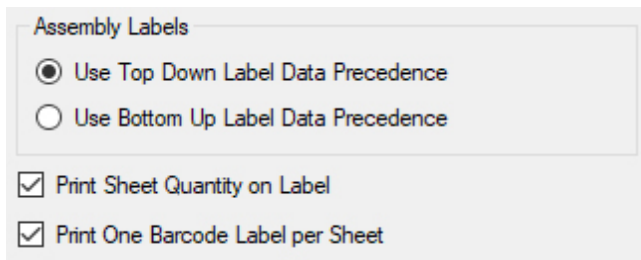
OutputSaw: If you have a CIM-Tech saw post processor, you can also select if you want to use one of your custom label setups for this option by checking the 'OutputSaw' box.



Active Label Configurations			
Active	Name	OutputTNP	OutputSaw
<input checked="" type="checkbox"/>	RCIM_DefaultLabeling	<input type="checkbox"/>	<input checked="" type="checkbox"/>

Assembly Labels

This options allows you to define how the labels will be generated depending on if assemblies/jobs are used instead of parts.



Assembly Labels

☒ Use Top Down Label Data Precedence

☐ Use Bottom Up Label Data Precedence

☒ Print Sheet Quantity on Label

☒ Print One Barcode Label per Sheet

Use Top Down Label Data Precedence: When this option is selected, the label information used to create the labels for the job will be based on the information included in the job that contains the assemblies. Information that is included in the assemblies/jobs that have been added to the master job will be ignored.

Use Bottom Up Label Data Precedence: When this option is selected, the label information used to create the labels for the job will be based on the information included in the assemblies/jobs that have been added to the master job. The information that is included in the master job will be ignored.

Print Sheet Quantity on Label

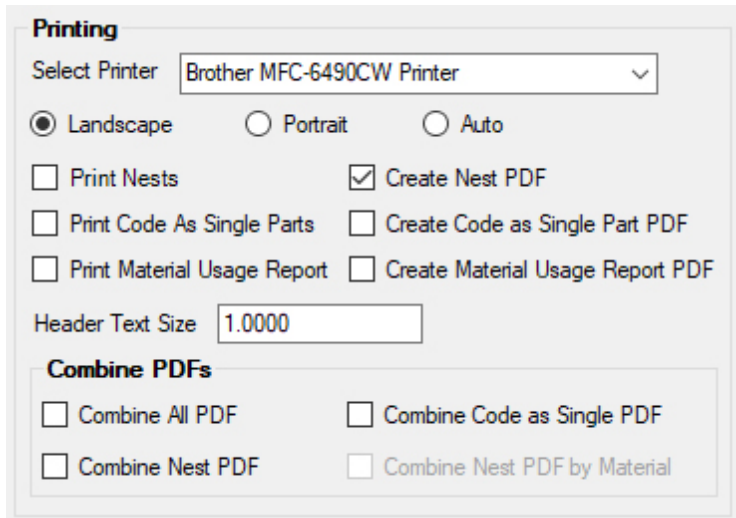
Turned on, you will have the number of the sheet listed for each NC code file on the label.

Print One Barcode Label per Sheet

If checked, there will be a barcode label printed in the label file for each sheet nested in the job.

Printing

This area allows you to select if you would like Router-CIM Automation Suite to create PDFs and/or print out certain items after job completion.



Printing

Select Printer

☒ Landscape ☐ Portrait ☐ Auto

☐ Print Nests ☒ Create Nest PDF

☐ Print Code As Single Parts ☐ Create Code as Single Part PDF

☐ Print Material Usage Report ☐ Create Material Usage Report PDF

Header Text Size

Combine PDFs

☐ Combine All PDF ☐ Combine Code as Single PDF

☐ Combine Nest PDF ☐ Combine Nest PDF by Material

Select Printer

This option will allow you to select a printer for the plotting of the nested and non-nested parts.

Printing Layout

Select one of the options for how the printed option is laid out on the sheet.

Print Nests

If checked, you will get a print of each nested sheet sent to the printer automatically during the job run. One print for each nest.

Print Code As Single Parts

If checked and you have parts that are set to Code as Single Part, they will be printed during the job run. One print for each part.

Print Material Usage Report

Turned on, a material report will automatically be printed at the end of the job.

Create Nest PDF

If checked, you will get a PDF of each nested sheet with the name of the PDF matching the output file named defined in the ['Output Files'](#) section.

Create Code as Single Part PDF

If checked, you will get a PDF of each Code Single Part (if applicable) with the name of the PDF matching the part name file named defined in the ['Part Name'](#) section.

Create Material Usage Report PDF

If checked, you will get a PDF of the material usage report.

Header Text Size

Use this field to adjust the text size of the header that is included at the top of each nest in the nested drawing file and code as single drawing file.

**Combine PDFs**

If you are creating PDFs, you have options on the included PDFs per file.

Combine All PDF

When this option is checked, all PDFs that were created will be combined into one file.

Combine PDFs	
<input checked="" type="checkbox"/> Combine All PDF	<input type="checkbox"/> Combine Code as Single PDF
<input type="checkbox"/> Combine Nest PDF	<input type="checkbox"/> Combine Nest PDF by Material

Combine Nest PDF

When this option is checked, all Nest PDFs that were created will be combined into one file or you can have them combined by Material.

All Nest PDFs combined as one file:

Combine PDFs

☐ Combine All PDF ☐ Combine Code as Single PDF

☒ Combine Nest PDF ☐ Combine Nest PDF by Material

Nest PDFs combined per material:

Combine PDFs

☐ Combine All PDF ☐ Combine Code as Single PDF

☒ Combine Nest PDF ☒ Combine Nest PDF by Material

Combine Code as Single PDF

When this option is checked, all Code as Single PDFs that were created will be combined into one file.

Combine PDFs

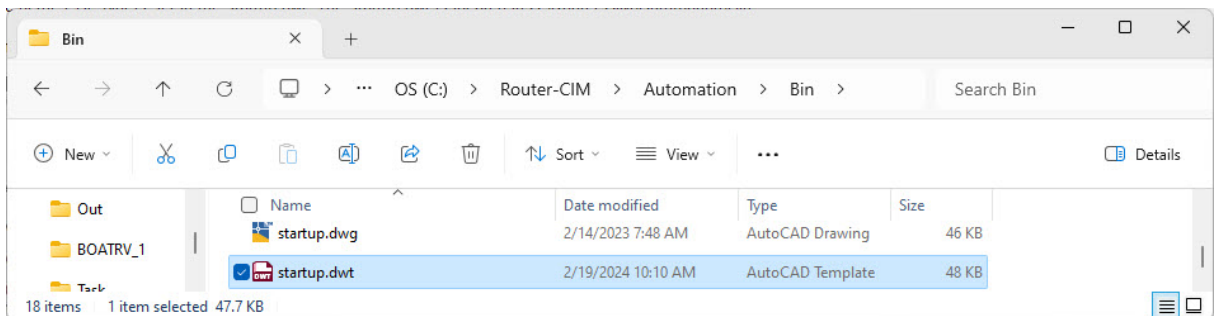
☐ Combine All PDF ☒ Combine Code as Single PDF

☐ Combine Nest PDF ☐ Combine Nest PDF by Material

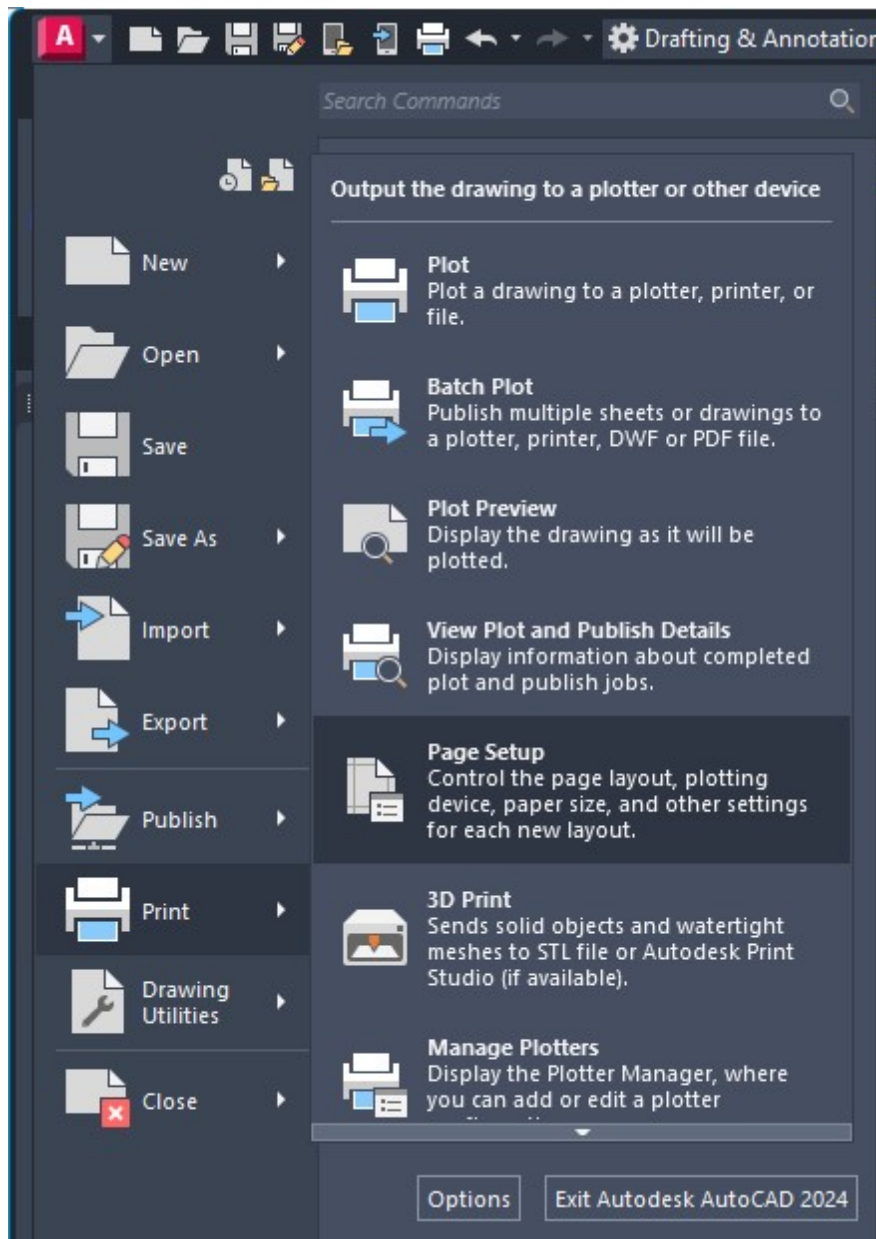
Changing PDF Plot Page Size

The size of the PDF plot is set in the startup.dwt. The startup.dwt is located in C:\Router-CIM\Automation\Bin.

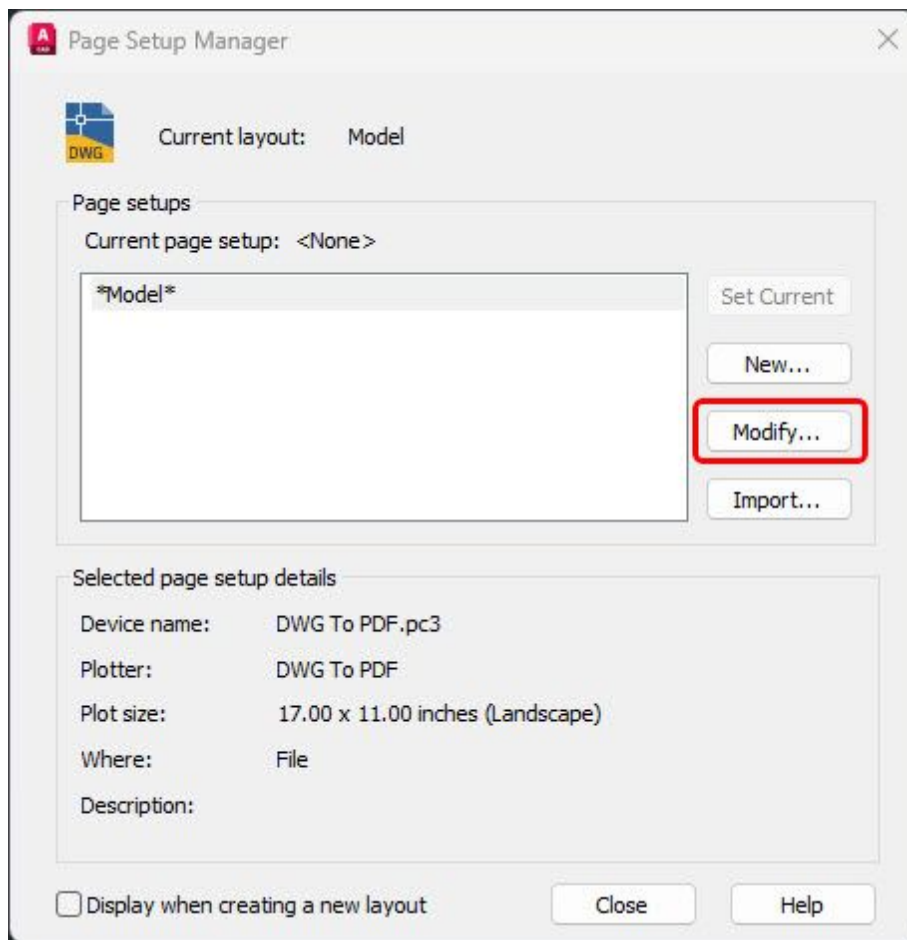
To change the size of the PDF plot, navigate in File Explorer to the C:\Router-CIM\Automation\Bin folder and open the startup.dwt file in AutoCAD.



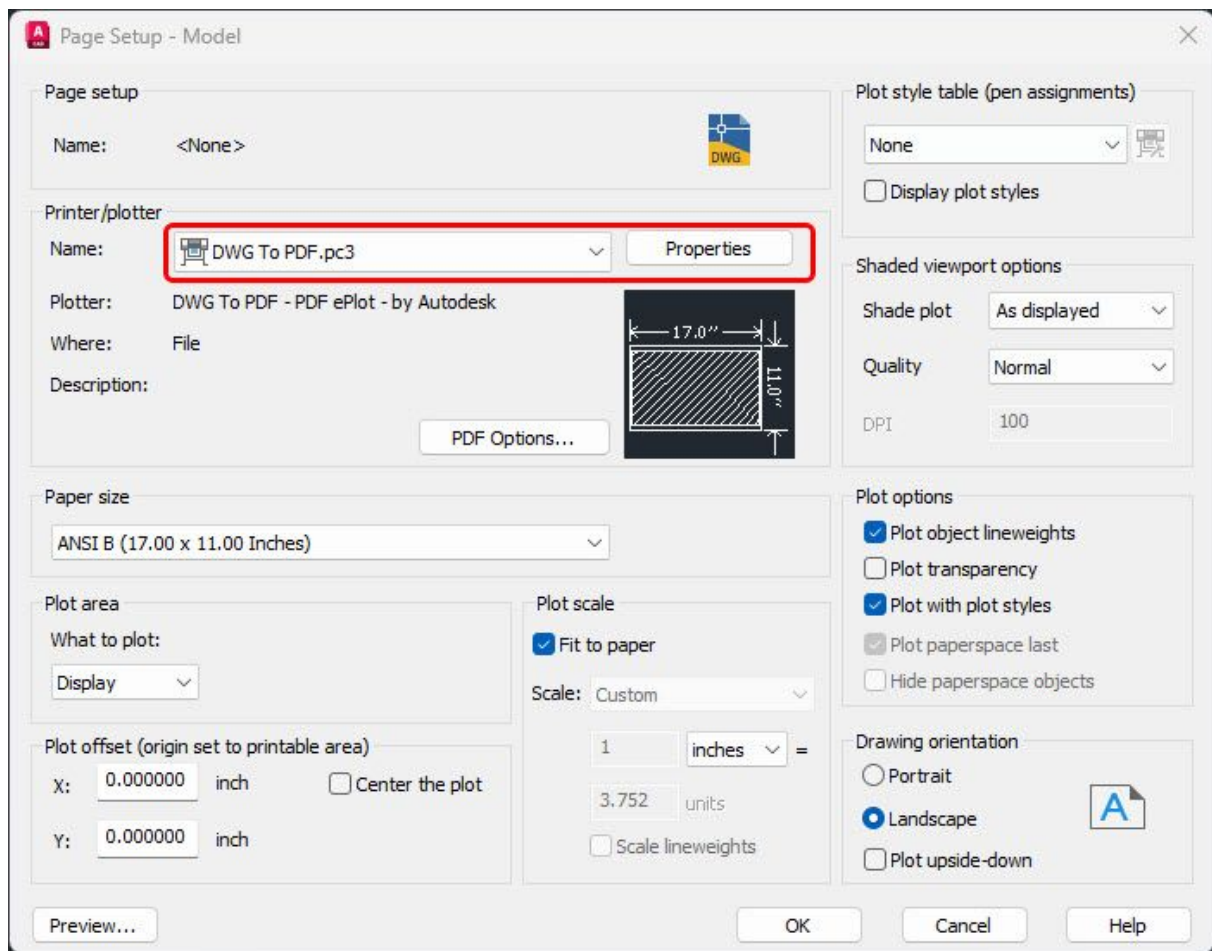
Once the startup.dwg opens in AutoCAD, go to the upper left and select on the drop-down, go to 'Print' and then select 'Page Setup'.



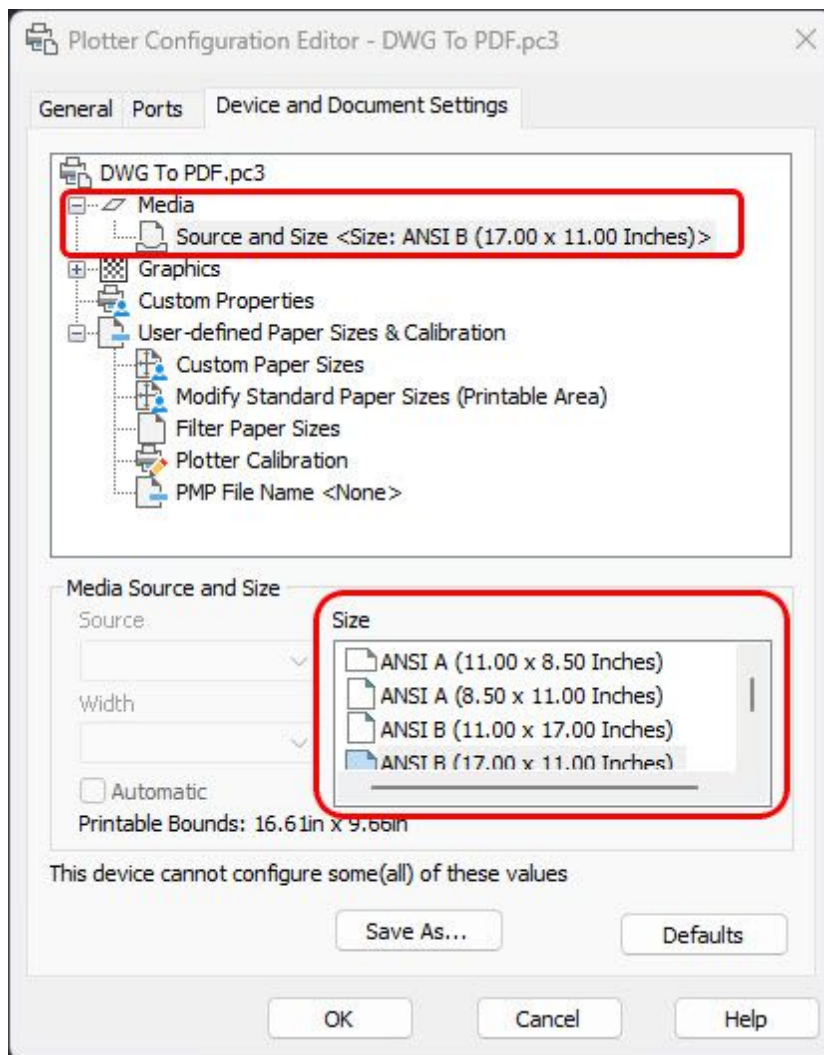
This will open the Page Setup Manager. Select the *Model* under Page Setups and then select 'Modify'.



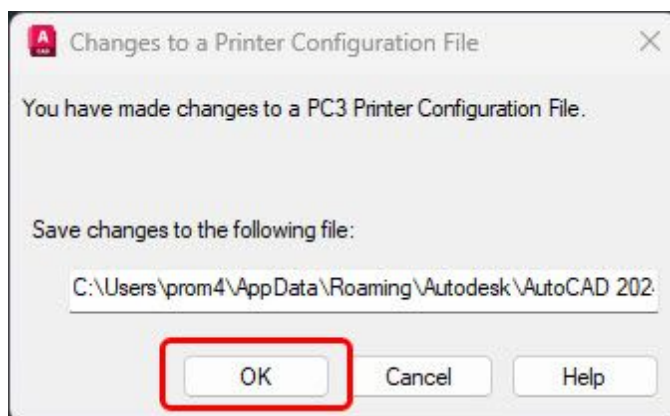
Once selected, the 'Page Setup - Model' screen will appear. Change the Printer/plotter using the drop-down to DWG to PDF.pc3 and then select 'Properties'.



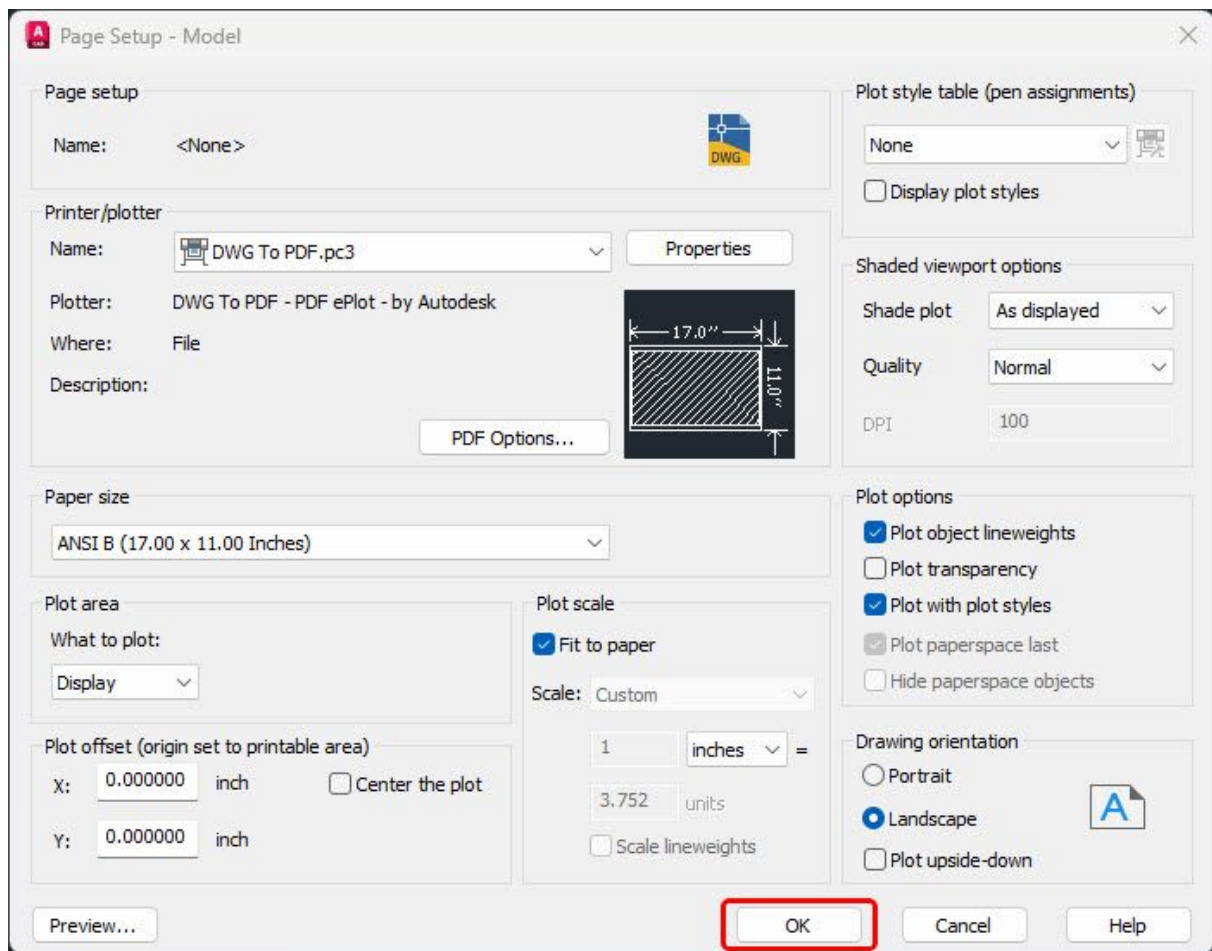
The 'Properties' screen will open. Expand the 'Media' section and select on 'Source and Size'. You will then be able to select the desired size plot within the 'Size' section shown here.



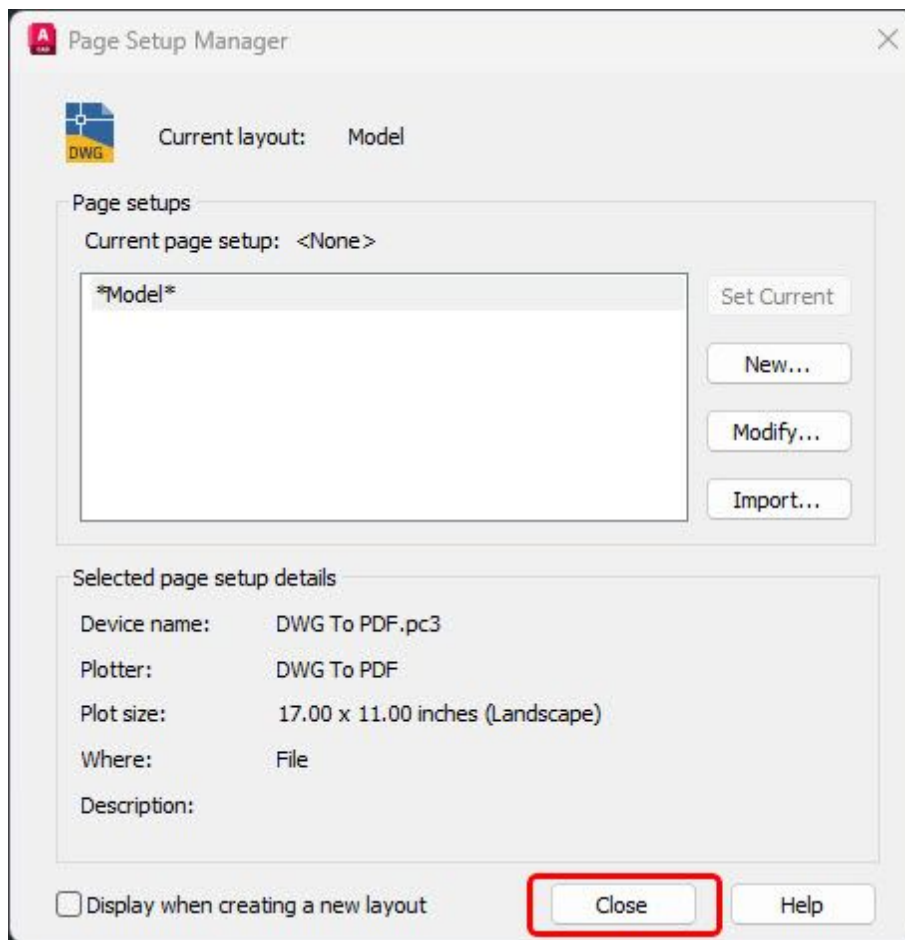
Once you have made the desired change of plot size, select 'OK'. This will then ask you to save the changes and select the 'OK' button.



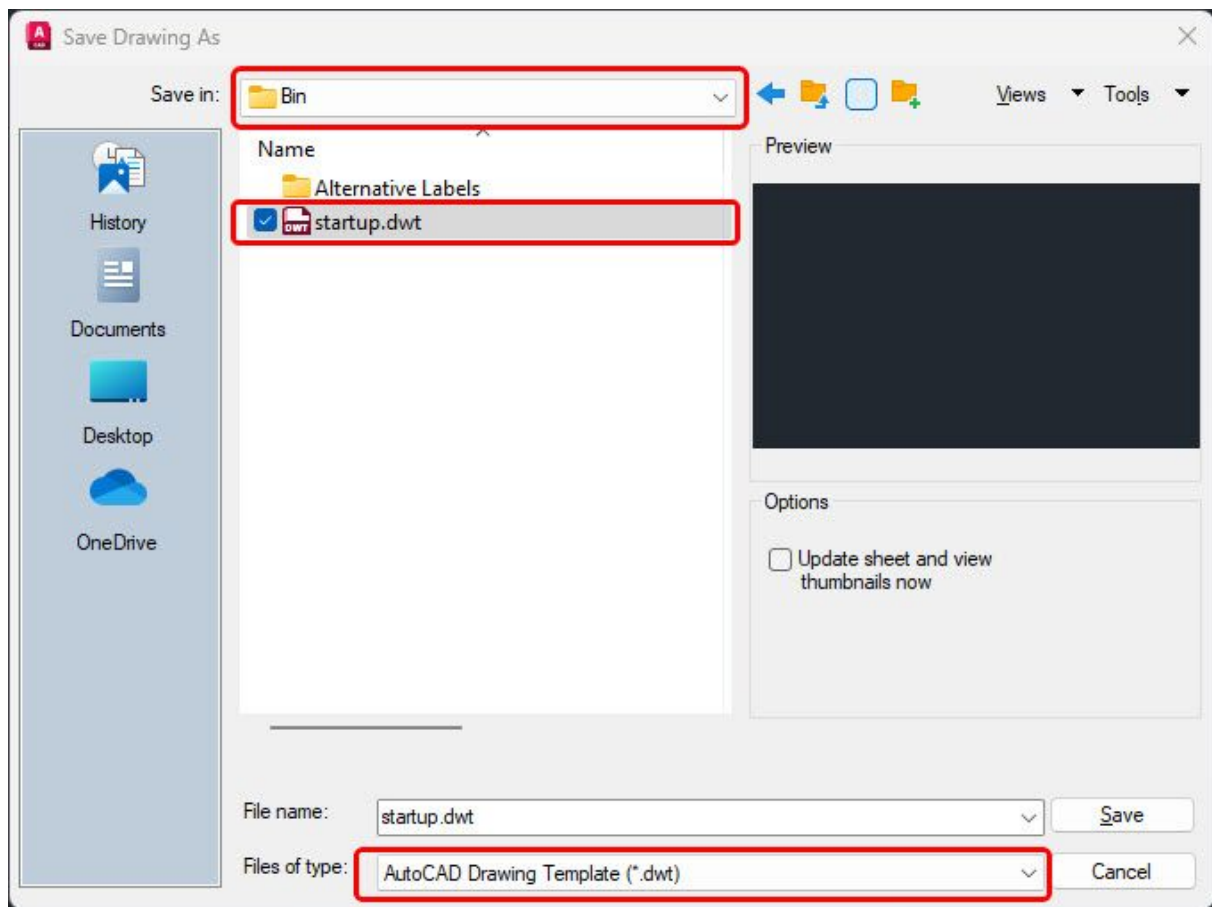
Select 'OK' on the Page Setup - Model to apply the changes.



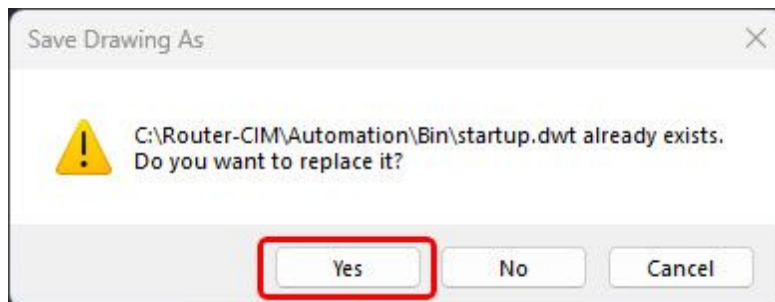
You can then select 'Close' on the 'Page Setup Manager'.



Once back in AutoCAD, do a 'Save As'. Change the 'Files of type:' to the AutoCAD Drawing Template (*.dwt). You will then navigate to C:\Router-CIM\Automation\Bin and select the 'startup.dwt'. Select the 'Save' button.



It will ask if you want to replace the file and select 'Yes'. You will then be able to close AutoCAD and start Router-CIM Automation Suite. The PDFs created will now be the size you selected.



Open Router-CIM Automation Suite and go to [Tools](#) and select [Restore Startup Drawing](#) to load the updated startup.dwt with your correct page size.

Reporting

After job run, you have multiple reporting options.

Create Nest Summary Report

When this option is checked, you will receive a report file that includes the code file names, tooling used, parts nested and additional information that can be used as a Job Sheet.

Reporting

☐ Reset Sheet Counter for Each Material

☒ Create Nest Summary Report

☐ Create Single Part Summary Report

Create Single Part Summary Report

When this option is checked, you will receive a report file for each individual part file including code file names, tooling used and additional information.

Reporting

☐ Reset Sheet Counter for Each Material

☒ Create Nest Summary Report

☒ Create Single Part Summary Report

Reset Sheet Counter for Each Material

If checked, the counter for the NC code file will have its numbers reset each time it encounters a new material. So if you had two nests, one on MDF75_ and one on BIR500_, you would get files named MDF75_1 and BIR500_1. If the box was unchecked, you would get MDF75_1 and BIR500_2. The last digit is the sheet count.

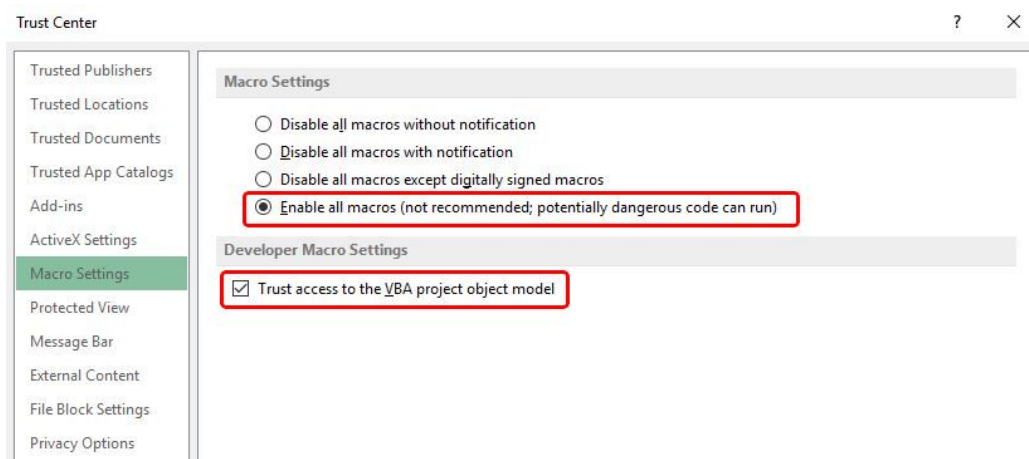
Reporting

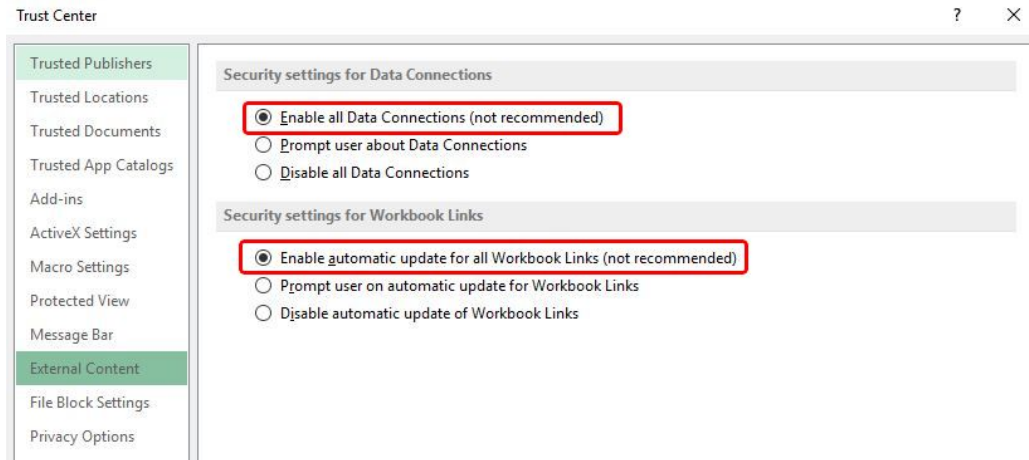
☒ Reset Sheet Counter for Each Material

☒ Create Nest Summary Report

☐ Create Single Part Summary Report

NOTE: Security settings in Excel may need to be adjusted in order to view the Summary Reports correctly. You will need to go to the Microsoft Excel 'Trust Center' located under the 'Options' in Microsoft Excel. To allow the Summary Report to populate, you will need to adjust the Macro Settings and the External Content settings as shown below.





Part Name

The Part Name tab is to set the name of the part as it shows in the nested sheets AND will be the name of the code file if 'Code as Single Part' option is being used. For the code files, they will default with beginning with the letter S. SSample1.out. If you wish to change the prefix or suffix of the single code files, please refer to the variable ['Output Files'](#) section.

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
<div> <div>Section 1</div> <div> <input type="radio"/> Part Name <input checked="" type="radio"/> Part Number </div> </div> <div> <div>Section 2</div> <div> <input type="radio"/> Part Name <input type="radio"/> Part Number <input type="radio"/> Material <input type="radio"/> X Dim <input type="radio"/> Y Dim <input type="radio"/> Z Dim <input type="radio"/> Quantity <input type="radio"/> Job Name <input type="radio"/> Label 1 <input type="radio"/> Label 2 <input type="radio"/> Label 3 <input type="radio"/> Label 4 <input checked="" type="radio"/> None </div> <div>Section Prefix</div> <div>—</div> </div> <div> <div>Section 3</div> <div> <input type="radio"/> Part Name <input type="radio"/> Part Number <input type="radio"/> Material <input type="radio"/> X Dim <input type="radio"/> Y Dim <input type="radio"/> Z Dim <input type="radio"/> Quantity <input type="radio"/> Job Name <input type="radio"/> Label 1 <input type="radio"/> Label 2 <input type="radio"/> Label 3 <input type="radio"/> Label 4 <input checked="" type="radio"/> None </div> <div>Section Prefix</div> <div>—</div> </div> <div> <div>Section 4</div> <div> <input type="radio"/> Part Name <input type="radio"/> Part Number <input type="radio"/> Material <input type="radio"/> X Dim <input type="radio"/> Y Dim <input type="radio"/> Z Dim <input type="radio"/> Quantity <input type="radio"/> Job Name <input type="radio"/> Label 1 <input type="radio"/> Label 2 <input type="radio"/> Label 3 <input type="radio"/> Label 4 <input checked="" type="radio"/> None </div> <div>Section Prefix</div> <div>—</div> </div> <div> <div>Section 5</div> <div> <input type="radio"/> Part Name <input type="radio"/> Part Number <input type="radio"/> Material <input type="radio"/> X Dim <input type="radio"/> Y Dim <input type="radio"/> Z Dim <input type="radio"/> Quantity <input type="radio"/> Job Name <input type="radio"/> Label 1 <input type="radio"/> Label 2 <input type="radio"/> Label 3 <input type="radio"/> Label 4 <input checked="" type="radio"/> None </div> <div>Section Prefix</div> <div>—</div> </div>								

Sample Part Name

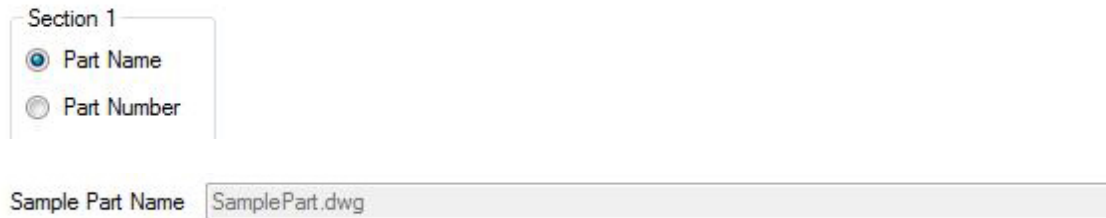
1

There are 5 possible sections to compile the data embedded in the part name as it appears on the nested sheet. Each section will appear in the **Sample Part Name** box at the bottom to show an example of how the name might look should certain options be specified.

The default is to use only the Part Number.

The prefix for each section can be specified as a character you wish displayed between each field.

The following shows Section 1 set to Part Name instead of Part Number:



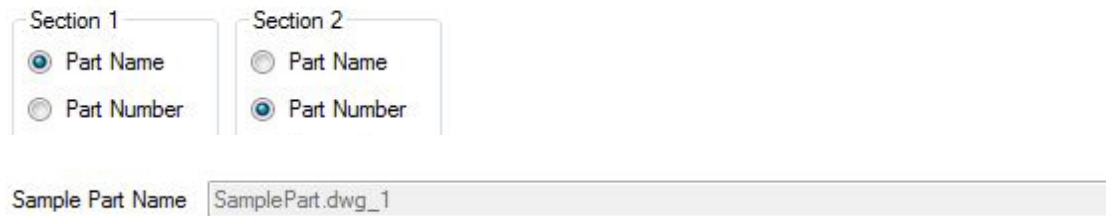
Section 1

☒ Part Name

☐ Part Number

Sample Part Name: SamplePart.dwg

Changing Section 1 to Part Number and Section 2 to Part Name gives you this result:



Section 1

☒ Part Name

☐ Part Number

Section 2

☐ Part Name

☒ Part Number

Sample Part Name: SamplePart.dwg_1

And using all 5 sections could give you the following result (which is a very long part name).

Section 1	Section 2	Section 3	Section 4	Section 5
<input checked="" type="radio"/> Part Name	<input type="radio"/> Part Name	<input type="radio"/> Part Name	<input type="radio"/> Part Name	<input type="radio"/> Part Name
<input type="radio"/> Part Number	<input checked="" type="radio"/> Part Number	<input type="radio"/> Part Number	<input type="radio"/> Part Number	<input type="radio"/> Part Number
	<input type="radio"/> Material	<input checked="" type="radio"/> Material	<input type="radio"/> Material	<input type="radio"/> Material
	<input type="radio"/> X Dim	<input type="radio"/> X Dim	<input type="radio"/> X Dim	<input type="radio"/> X Dim
	<input type="radio"/> Y Dim	<input type="radio"/> Y Dim	<input type="radio"/> Y Dim	<input type="radio"/> Y Dim
	<input type="radio"/> Z Dim	<input type="radio"/> Z Dim	<input type="radio"/> Z Dim	<input type="radio"/> Z Dim
	<input type="radio"/> Quantity	<input type="radio"/> Quantity	<input checked="" type="radio"/> Quantity	<input type="radio"/> Quantity
	<input type="radio"/> Job Name	<input type="radio"/> Job Name	<input type="radio"/> Job Name	<input type="radio"/> Job Name
	<input type="radio"/> Label 1	<input type="radio"/> Label 1	<input type="radio"/> Label 1	<input checked="" type="radio"/> Label 1
	<input type="radio"/> Label 2	<input type="radio"/> Label 2	<input type="radio"/> Label 2	<input type="radio"/> Label 2
	<input type="radio"/> Label 3	<input type="radio"/> Label 3	<input type="radio"/> Label 3	<input type="radio"/> Label 3
	<input type="radio"/> Label 4	<input type="radio"/> Label 4	<input type="radio"/> Label 4	<input type="radio"/> Label 4
	<input type="radio"/> None	<input type="radio"/> None	<input type="radio"/> None	<input type="radio"/> None
	Section Prefix -	Section Prefix -	Section Prefix -	Section Prefix -

Sample Part Name

Keep in mind that this is to set the name of the part as it shows in the nested sheets. If you have small or narrow parts, and a lot of combinations to the name field, the text will overlap other parts and could become confusing.

To manually set a point for the part name to be placed on individual parts, you will need to add the variable `*nestpoint*` in the NCVAR file. Please go the [Common Variables in Automation](#) section to find out how to add this variable.

Once the variable has been entered and the value set to **T**, you will need to place an AutoCAD "point" on the part drawing itself in the location that you want the part name to be located. This "point" will need to be on the layer **NESTCENTER** that you create in the part's AutoCAD drawing. When labeling the part, Router-CIM Automation Suite will insert the part name at the "point" location you defined on layer **NESTCENTER**.

For information on how to insert points through AutoCAD, please refer to the AutoCAD Help file.

Output Files

This is the location where you can specify the file extension that you need the NC code file to be. Simply change the 'out' to your required extension. All NC code output files will have this extension.

Output File Extension

Output File Name

Similar to the Part Name section the Output Files parameters contain 5 sections to allow you to string together names for the Code files that have the proper meaning for you.

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results	
Output File Extension		<input type="text" value="OUT"/>		Code as Single Prefix		<input type="text" value="S"/>		Code as Single Suffix	<input type="text"/>
Section 1 <input checked="" type="radio"/> Job Name <input type="radio"/> Material Code <input type="radio"/> First Material Attribute <input type="radio"/> Label Field 1 <input type="radio"/> Label Field 2		Section 2 <input type="radio"/> Job Name <input type="radio"/> Material Code <input checked="" type="radio"/> First Material Attribute <input type="radio"/> Label Field 1 <input type="radio"/> Label Field 2 <input type="radio"/> None		Section 3 <input type="radio"/> Job Name <input checked="" type="radio"/> Material Code <input type="radio"/> First Material Attribute <input type="radio"/> Label Field 1 <input type="radio"/> Label Field 2 <input type="radio"/> None		Section 4 <input type="radio"/> Job Name <input type="radio"/> Material Code <input checked="" type="radio"/> First Material Attribute <input type="radio"/> Label Field 1 <input type="radio"/> Label Field 2 <input type="radio"/> None		Section 5 <input type="radio"/> Job Name <input type="radio"/> Material Code <input type="radio"/> First Material Attribute <input type="radio"/> Label Field 1 <input type="radio"/> Label Field 2 <input checked="" type="radio"/> None	
<input type="text" value="BOATRV_1BlueMDF75Blue"/>									

The default name for the code files is set to Material Name. Using other combinations appends those combinations to the name. Examples of the name are show below.

Using all 5 fields can create a very long filename, and there are no delimiters between the sections (as some controllers will not allow special characters).

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results	
Output File Extension		<input type="text" value="OUT"/>		Code as Single Prefix		<input type="text" value="S"/>		Code as Single Suffix	<input type="text"/>
Section 1 <input type="radio"/> Job Name <input checked="" type="radio"/> Material Code <input type="radio"/> First Material Attribute <input type="radio"/> Label Field 1 <input type="radio"/> Label Field 2		Section 2 <input checked="" type="radio"/> Job Name <input type="radio"/> Material Code <input type="radio"/> First Material Attribute <input type="radio"/> Label Field 1 <input type="radio"/> Label Field 2 <input type="radio"/> None		Section 3 <input type="radio"/> Job Name <input type="radio"/> Material Code <input checked="" type="radio"/> First Material Attribute <input type="radio"/> Label Field 1 <input type="radio"/> Label Field 2 <input type="radio"/> None		Section 4 <input type="radio"/> Job Name <input type="radio"/> Material Code <input type="radio"/> First Material Attribute <input checked="" type="radio"/> Label Field 1 <input type="radio"/> Label Field 2 <input type="radio"/> None		Section 5 <input type="radio"/> Job Name <input type="radio"/> Material Code <input type="radio"/> First Material Attribute <input type="radio"/> Label Field 1 <input checked="" type="radio"/> Label Field 2 <input type="radio"/> None	
<input type="text" value="MDF75BOATRV_2BlueJobLabel1JobLabel2"/>									

So this filename would be MDF7512_Boat_Parts_BlueJobLabel1JobLabel2.out

Code as Single Prefix and Suffix

For the code files, they will default with beginning with the letter S. SSample1.out. If you wish to change the prefix or suffix of the single code files, these fields allow you to make the change.

Default Prefix

SSample1.OUT

Code as Single Prefix Code as Single Suffix

Change Prefix
S_Sample1.OUT

Code as Single Prefix Code as Single Suffix

Change Suffix
Sample1_S.OUT

Code as Single Prefix Code as Single Suffix

Results

The results section will display a log for each time the job is run, and place in the log the results of the run.

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results
Entry Type	Entry Date	Entry Text						
▶ 3	09/16/2016	Processing has started.						
3	02/17/2016	Processing has started.						
3	09/27/2016	Processing has started.						
3	09/27/2016	Processing has started.						
3	09/27/2016	Job BOATRV_1_CODE_SINGLE_PART Completed Successfully. Elapsed time: 2 Minutes, 10 Seconds						
3	02/17/2016	Job CopyOf_BOATRV_1 Completed Successfully. Elapsed time: 2 Minutes, 0 Seconds						
3	09/27/2016	Job BOATRV_1_CODE_SINGLE_PART Completed Successfully. Elapsed time: 2 Minutes, 10 Seconds						
3	09/16/2016	Job BOATRV_1_CODE_SINGLE_PART Completed Successfully. Elapsed time: 2 Minutes, 30 Seconds						

You can double-click on any of these logs, and you will see the summary:

Log Entry

Date: 1/22/2009

Type:

Description: Job CartControl_1 Completed Successfully. Elapsed time: 2 minutes and 45 seconds

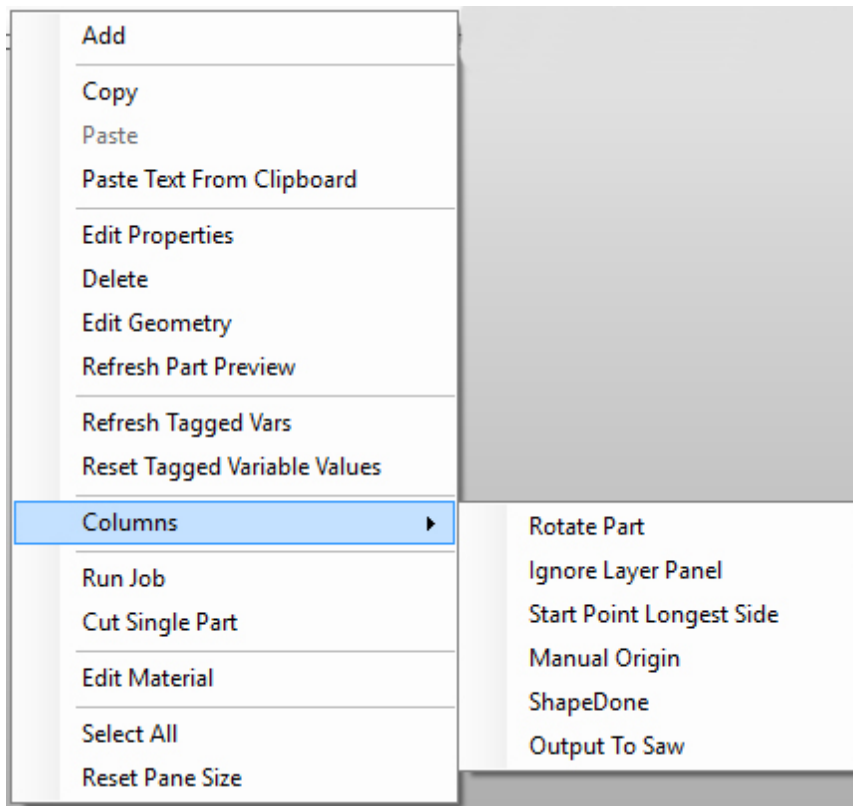
OK Cancel

Also right-clicking on an entry shows a menu which will allow you to add a new entry, edit an entry, or delete an entry.

	Entry Type	Entry Date	Entry Text
■	Audit Entry	1/22/2022	Completed Successfully. Elapsed time: 2 minutes and 45 seconds
■	Audit Entry	1/22/2022	Completed Successfully. Elapsed time: 2 minutes and 41 seconds
■	Audit Entry	1/22/2022	Completed Successfully. Elapsed time: 2 minutes and 55 seconds

Part Window Right Click Options

Router-CIM Automation Suite offers right click options for the parts in the currently selected job.



Add

Selecting this option will open the Add Parts window. This window has several options, you can select default folders rcim_work, or Router-CIM as well as set the default types of files to DWG, DXF, or SCN files. You can see a preview of the part, unless multiple parts are selected.

Copy

Selecting this option will copy the part/multiple parts from the parts list.

Paste

Selecting this option will paste the part/multiple parts and add them to the bottom of the parts list.

Paste Text from Clipboard

Selecting this option will paste any text that is currently held in your clipboard.

Edit Properties

This selection opens the part properties window.

For more information go to the [Part Properties](#) section.

Delete

Selecting this option will delete the selected part from within a job.

Edit Geometry

Selecting Edit Geometry will open AutoCAD and show the currently selected part opened from its original location. If no part is selected, you will be prompted to select a part first.

Refresh Part Preview

Selecting Refresh Part Preview will regenerate the preview of the part in the Part Previewer.

For more information go to the [Part Previewer](#) section.

Refresh Tagged Variables

Selecting Refresh Tagged Variables will regenerate the tagged variables on a Macro part (SCN).

For more information go to the [Tagged Variables](#) section.

Reset Tagged Variables Values

Selecting this option will reset the tagged variables of a macro back to their original values.

Columns

This allows you to add additional columns to the Part Window to show basics of some of the part properties.

Rotate Part - Shows if 'Rotate Part' is checked under the part properties

Ignore Layer Panel - Shows if 'Ignore Layer Panel' is checked under the part properties

Start Point Longest Side - Shows if 'Start Point Longest Side' is checked under the part properties

Manual Origin - Shows if 'Manual Origin' is checked under the part properties (Macros only)

ShapeDone - Shows if 'ShapeDone' is checked under the part properties

Output to Saw - Shows if 'Output to Saw' is checked under the part properties

Parts	Job Settings	Nesting	Advanced Nesting	Dynamic Variables	Printing and Labels	Part Name	Output Files	Results	IgnorePanel	ManualOrigin	RotatePart	StartPointLong	ShapeDone
1	Sample1.dwg	4	MATERIAL_0.5000			C:\rcim_work\Automation\Samples\DivGS\Sam			<input type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
2	Sample2.dwg	1	MATERIAL_0.5000			C:\rcim_work\Automation\Samples\DivGS\Sam			<input type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
3	Sample3.dwg	2	MATERIAL_0.5000			C:\rcim_work\Automation\Samples\DivGS\Sam			<input type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
4	Sample4.dwg	2	MATERIAL_0.5000			C:\rcim_work\Automation\Samples\DivGS\Sam			<input type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
5	Sample5.dwg	4	MATERIAL_0.5000			C:\rcim_work\Automation\Samples\DivGS\Sam			<input type="checkbox"/>		<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Run Job

This option starts the selected job in Router-CIM Automation Suite and begins processing.

Cut Single Part

This option will process only the currently selected part.

Edit Material

This option will open up the currently selected part's material in the Material Editor.

For more information go to the [Material Properties](#) section.

Select All

This option will select all the parts in the current job.

Reset Pane Size

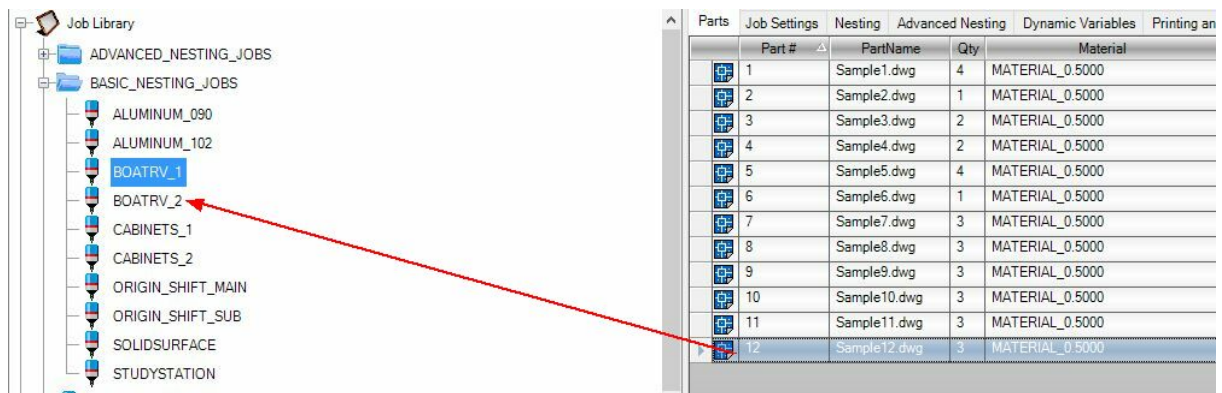
This option will reset the pane size of the Router-CIM Automation Suite window to the defaults.

Drag and Drop Parts

Router-CIM Automation Suite supports the drag and drop feature for copying parts from job to job.

Simply select the part you want to copy. Once the part is selected, left click on your mouse and hold. You will then be able to copy the part to another job.

For example if you wanted to copy the part Sample12.dwg from BOATRV_1 to BOATRV_2, select the Sample12.dwg in BOATRV_1. Once the part is selected, left click on your mouse and hold to drag the part into BOATRV_2.



Release the left mouse button and the part will be copied into BOATRV_2.

Note:

The screenshot displays the Router-CIM Automation Suite interface. On the left, the 'Job Library' tree shows a hierarchy of jobs, with 'BASIC_NESTING_JOBS' expanded to show various part types like 'ALUMINUM_090', 'BOATRV_1', 'BOATRV_2', 'CABINETS_1', 'CABINETS_2', 'ORIGIN_SHIFT_MAIN', 'ORIGIN_SHIFT_SUB', 'SOLIDSURFACE', 'STUDYSTATION', and code-based parts. The 'BOATRV_2' job is selected. Below the tree, a text box states: 'Sample job showing standard parts using the Basic Nesting add-on.'

On the right, the 'Parts' table lists 41 parts. The table has columns: Part #, PartName, Qty, Material, and Full. The parts are listed in descending order of quantity. Part 41, 'Sample12.dwg' with a quantity of 3 and material 'MATERIAL_0.5000', is highlighted with a red box.

At the bottom, the 'Knowledge and Post Info' panel is visible. It includes sections for 'Primary Knowledge' (WOOD_SCIM_RS.dwg), 'Secondary Knowledge' (WOOD_SCIM_RS.dwg), 'Primary Machine' (RS274D.\$PP), and 'Secondary Machine' (RS274D.\$PP). There are buttons for 'New', 'Edit', and 'Use Primary Knowledge for Secondary Operations'.

Part #	PartName	Qty	Material	Full
16	Sample16.dwg	6	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
17	Sample17.dwg	8	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
18	Sample18.dwg	2	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
19	Sample19.dwg	2	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
20	Sample20.dwg	4	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
21	Sample21.dwg	6	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
22	Sample22.dwg	4	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
23	Sample23.dwg	2	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
24	Sample24.dwg	2	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
25	Sample25.dwg	2	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
26	Sample26.dwg	1	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
27	Sample27.dwg	1	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
28	Sample28.dwg	1	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
29	Sample29.dwg	4	MAPLE_PLYWOOD_0.5000	C:\rcim_work\Autom
30	Sample30.dwg	4	MAPLE_PLYWOOD_0.5000	C:\rcim_work\Autom
31	Sample31.dwg	1	MAPLE_PLYWOOD_0.5000	C:\rcim_work\Autom
32	Sample32.dwg	1	MAPLE_PLYWOOD_0.5000	C:\rcim_work\Autom
33	Sample33.dwg	1	MAPLE_PLYWOOD_0.5000	C:\rcim_work\Autom
34	Sample34.dwg	12	MAPLE_PLYWOOD_0.5000	C:\rcim_work\Autom
35	Sample35.dwg	12	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
36	Sample36.dwg	18	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
37	Sample37.dwg	4	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
38	Sample38.dwg	8	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
39	Sample39.dwg	1	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
40	Sample40.dwg	2	BALTIC_BIRCH_PLYWOOD_0.5000	C:\rcim_work\Autom
41	Sample12.dwg	3	MATERIAL_0.5000	C:\rcim_work\Autom

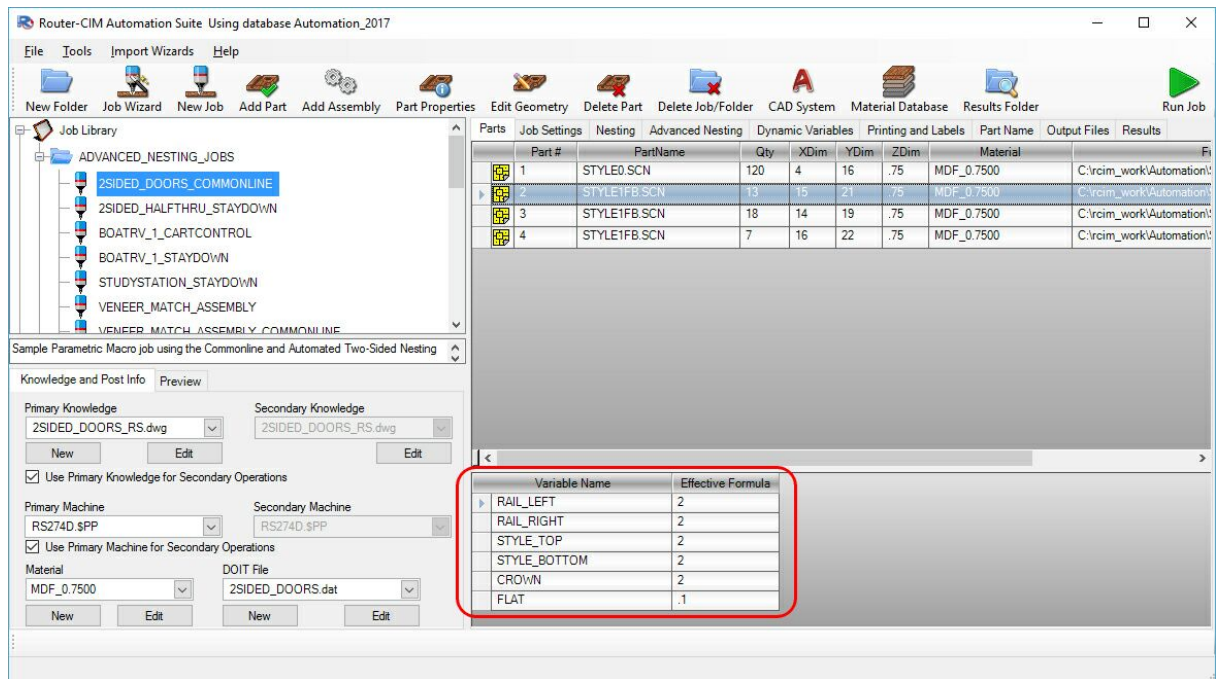
Variables Window

The variables window displays all tagged variables for a part, or any global variables for an assembly.

For more information on Tagged and Global Variable in Macros, click [here](#).

Tagged Variables

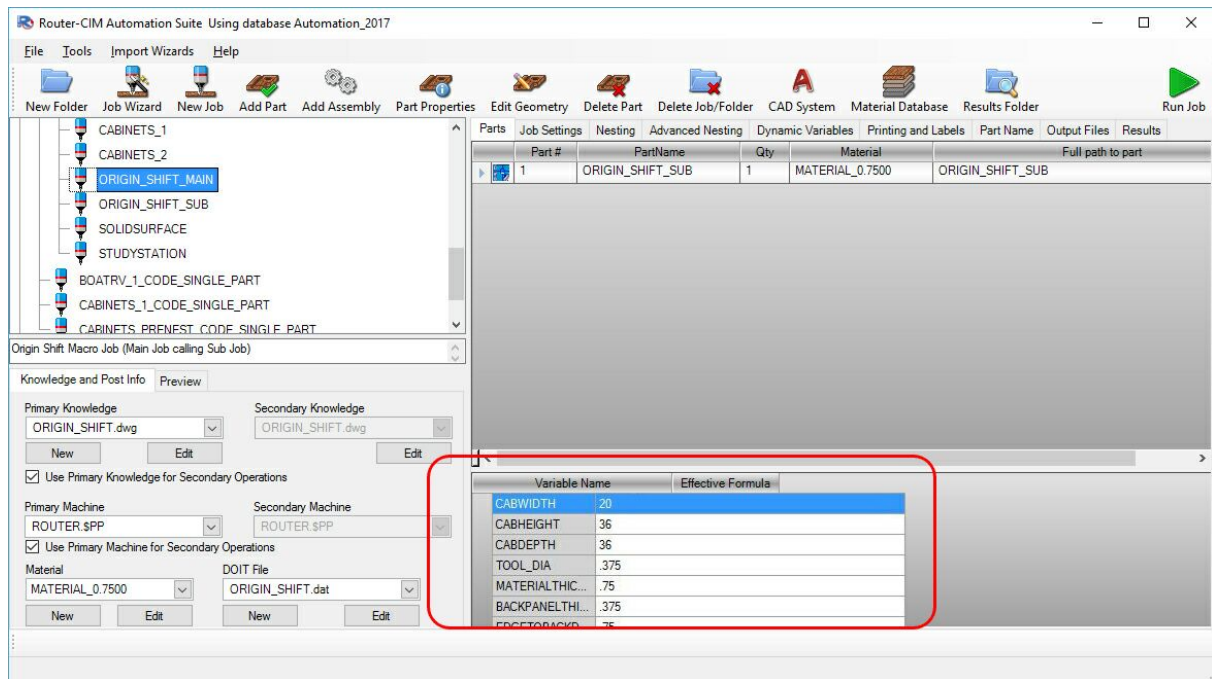
These variables are marked in the macro as tagged so that they may be modified in the job individually. In this instance a door macro is selected and several variables display in the Variables Window.



You can edit the Formula value for the tagged value by selecting it and typing in a new formula or value. You cannot change the variable name.

Global Variables

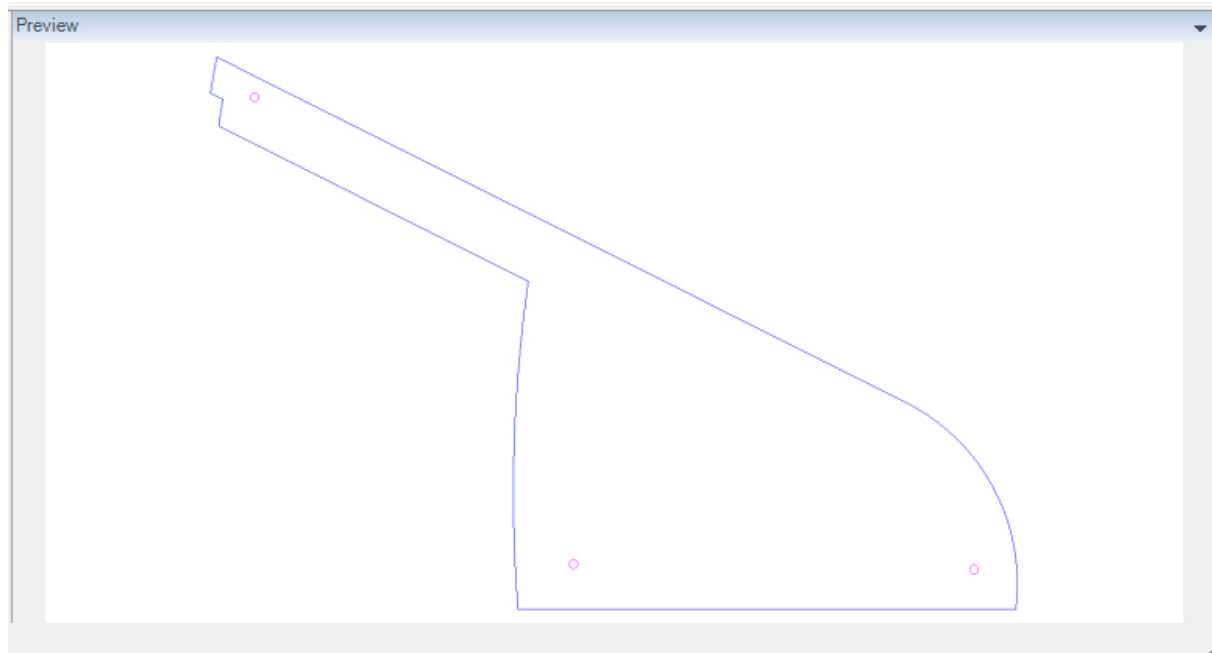
Global variables are typically variables that affect more than one part in an assembly. Instead of having the same variable in each part and trying to remember to change the value for each part, you would make a global variable and have it changed once and that change would affect each macro that uses that variable. For instance, there may be several macros that make up a cabinet, but you would only want to change the cabinets Height once and have all the parts relate to that value. Height would be a global variable.



This example has an assembly in it. The global variables for CABWIDTH, CABHEIGHT, and CABDEPTH can be changed to allow 3 different size cabinets, just from changing those 3 global variables. This way none of the parts in the sub job has to be changed on its own.

Part Preview Window

The part preview window will allow you to see a thumbnail image of the currently selected part. You can also suppress the display of the preview by un-selecting the **'Show Part Preview'** box by going to File/Settings under the General Settings tab.



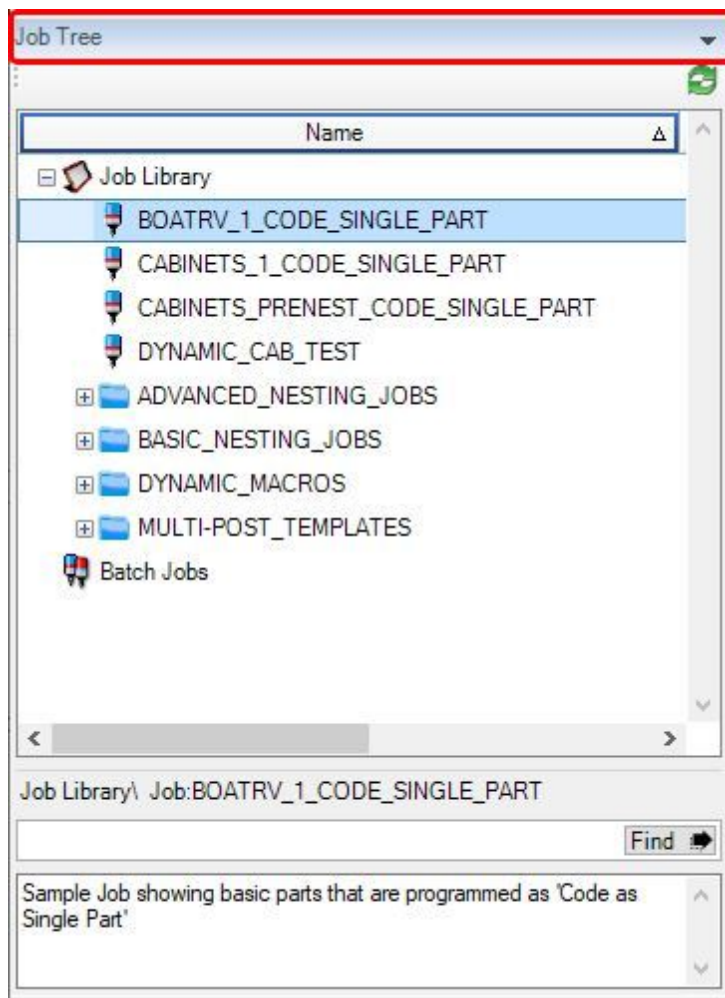
Moving Windows in Router-CIM Automation Suite

There are moveable windows in Router-CIM Automation Suite.

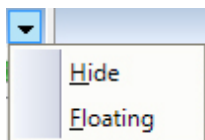
The following areas can be moved, floating or docked to customize the user interface to your liking:

- 1) Job Tree
- 2) Job Settings
- 3) Knowledge Post Info
- 4) Tagged Vars
- 5) Preview

Each moveable window can be identified with the light blue banner at the top of each window.



To the right of each banner is a drop down arrow allowing you to Hide or Float the window:

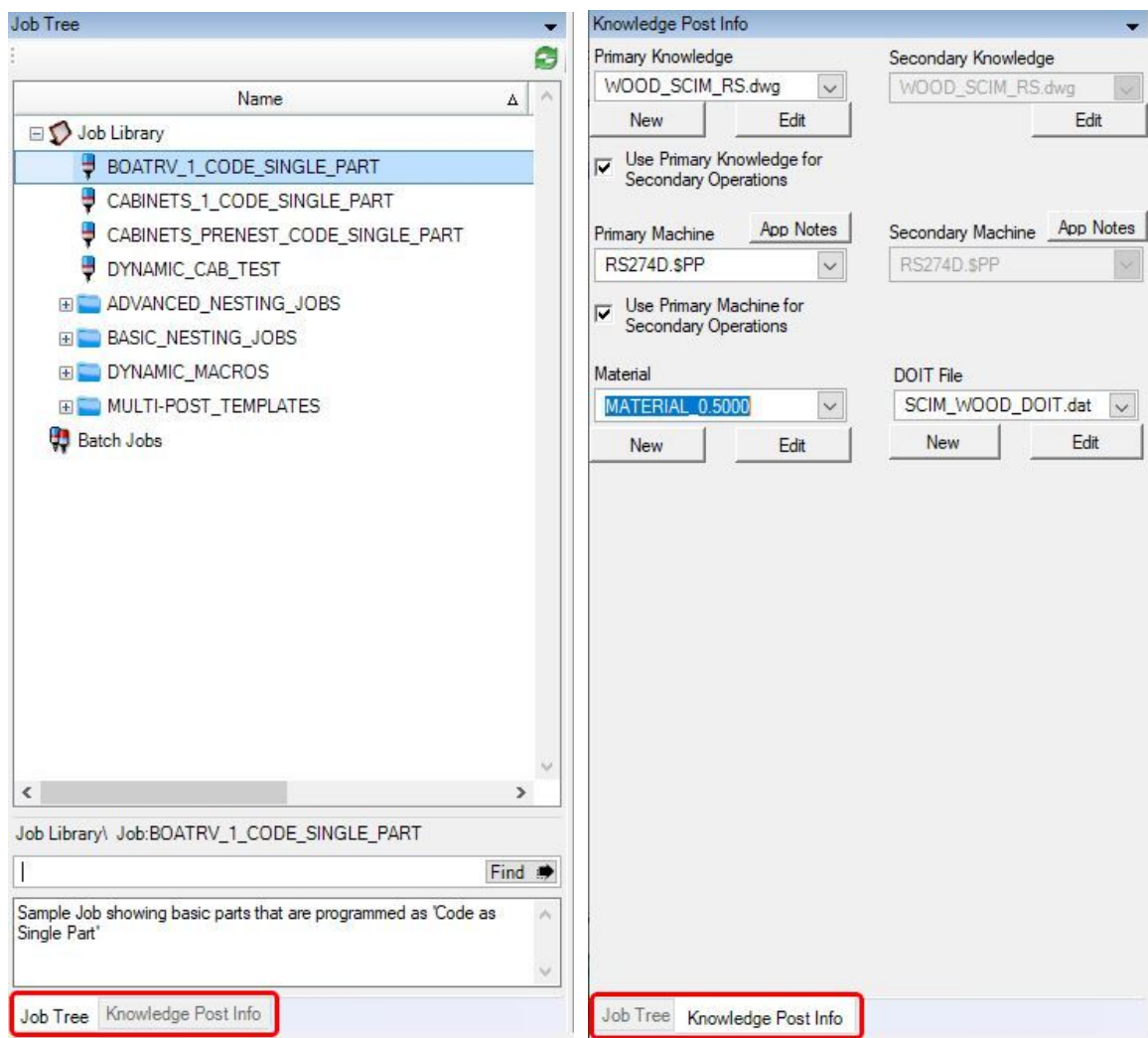


Floating will allow you to move the selected window within the Router-CIM Automation Suite or on to a separate monitor if you are running multiple monitors.

If you hide the window, it will not appear anymore in Router-CIM Automation Suite.

To bring a hidden window back, go to the **'Tools'** drop down and select **'Reset to Default Automation View'**.

As you move the window around, you will notice that it will show you the options for docking the window in a different location within the Router-CIM Automation Suite interface. Windows can be nested together and you will have a tab at the bottom of the nested windows to allow you to change in between the nested windows.



Knowledge Settings

Knowledge Settings are all the parameters relating to the Knowledge Drawings, DOIT file, Primary and Secondary Machines and Materials.

Knowledge and Post Info Preview

Primary Knowledge: WOOD_SCIM_RS.dwg New Edit

Secondary Knowledge: WOOD_SCIM_RS.dwg Edit

☒ Use Primary Knowledge for Secondary Operations

Primary Machine: RS274D.\$PP New Edit

Secondary Machine: RS274D.\$PP Edit

☒ Use Primary Machine for Secondary Operations

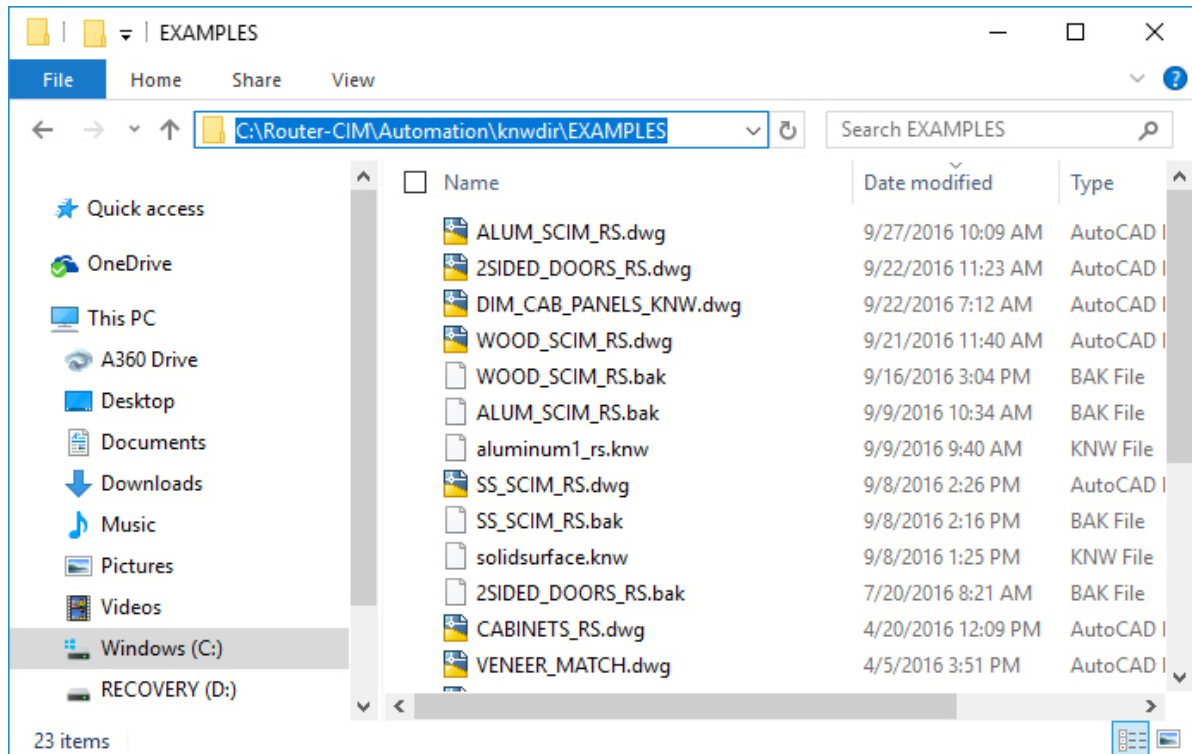
Material: MATERIAL_0.5000 New Edit

DOIT File: SCIM_WOOD_DOIT.dat New Edit

Primary Knowledge

For more information on your Knowledge Drawing, go to ['Edit Knowledge Drawing'](#).

The Primary Knowledge drawing is the drawing that contains the cut knowledges for the parts in a job. If each of the parts are to be nested together, then a single knowledge drawing is all that is necessary. By default, the knowledge drawings are stored in a sub-folder of Router-CIM. This location is editable by going to File/Settings under the [System Folders](#) tab.



New Knowledge Drawing

This button will allow a new Router-CIM session to start in AutoCAD. The post processor can be specified in the Configuration wizard. Then, new knowledges can be inserted and the drawing can be saved with a new name so that it is added to the list of available drawings.

NOTE: Once you have completed adding your knowledges in a new knowledge drawing, you **MUST** use the 'Save As' option in AutoCAD and name the file appropriately.

Edit Knowledge Drawing

Edit Drawing will allow you to have the Knowledge drawing opened in AutoCAD, with Router-CIM running so that you can add, change, or delete cut knowledges or edit fixtures or table drawings in the knowledge drawing that you have selected. Router-CIM will load with the same post processor configuration that the knowledge drawing was made with originally. For instance, if the drawing was configured for a specific post processor when it was made, then it will load into Router-CIM with that post processor in use when selected for editing.

Use Primary Knowledge for Secondary Operations

This allows you to use the same knowledge drawing for any secondary drawings that are to be cut, like a backside operation that has a separate drawing and needs separate code.

The image displays two screenshots of the 'Knowledge and Post Info' dialog box in the Router-CIM software. Both screenshots show the 'Primary Knowledge' dropdown set to 'WOOD_SCIM_RS.dwg' and the 'Secondary Knowledge' dropdown also set to 'WOOD_SCIM_RS.dwg'. Buttons for 'New', 'Edit', and 'Preview' are visible. In the top screenshot, the checkbox 'Use Primary Knowledge for Secondary Operations' is checked and highlighted with a red rectangle. In the bottom screenshot, the same checkbox is unchecked, and the 'Secondary Knowledge' dropdown menu and its 'Edit' button are highlighted with a red rectangle.

Secondary Knowledge

If a part in the job that is being processed has a backside operation that must be performed, a separate piece of NC Code must be made for that operation. In this instance, a Secondary Knowledge Drawing and Post processor can be used. When you unselect the box marked "Use Primary Drawing for Secondary Operations" another set of choices appears for a Secondary Knowledge Drawing and Post Processor.

Primary and Secondary Machine

The Primary and Secondary post processors may be selected from this list of available post processors. Only the post processors in the working Router-CIM\Ncpost folder will be available for selection. If you do not see the post processor for your machine in this list, contact CIM-Tech for more information.

If the Secondary machine selection is not available, uncheck the box marked Use Primary Machine for Secondary Operations.

The image shows two screenshots of a software interface. The top screenshot shows two dropdown menus, 'Primary Machine' and 'Secondary Machine', both set to 'RS274D.\$PP'. Below them is a checkbox labeled 'Use Primary Machine for Secondary Operations' which is checked. A red rectangle highlights this checkbox. The bottom screenshot shows the same two dropdown menus, both set to 'RS274D.\$PP', but the checkbox is unchecked. A red rectangle highlights the 'Secondary Machine' dropdown menu.

DOIT File

Router-CIM Automation Suite runs on the drawing Layer to Cut Knowledge association principle. That means that for each Layer in a drawing that has geometry that you want cut on the machine, a Cut Knowledge must exist. Further, the two of them must be associated together in the system to allow Router-CIM Automation Suite to know that the partnership exists. From this area you may select a DOIT file (that is a Layer/Knowledge Association file) from the list of available DOIT files in the \Router-CIM\Automation\Doitdir folder. You can edit the selected file and add/remove associations, and even create new DOIT files for use in Router-CIM Automation Suite.

NOTE: If you are creating a DOIT file with the 'New' button, you MUST use the 'Save As' option and name the file appropriately through the DOIT Editor.

For more information on DOIT, go to ['Edit DOIT File'](#).

The image shows a dialog box titled 'DOIT File'. It contains a dropdown menu with 'SCIM_WOOD_DOIT.dat' selected. Below the dropdown are two buttons: 'New' and 'Edit'.

Material

You can select the jobs material here from the list of available materials in the materials database. You can also create or edit a material with the options provided.

The image shows a dialog box titled 'Material'. It contains a dropdown menu with 'MATERIAL_0.5000' selected. Below the dropdown are two buttons: 'New' and 'Edit'.

Knowledge Drawing

A knowledge drawing is where your knowledges (ways to cut) are stored. This will be an AutoCAD drawing where each knowledge is stored as a block that Router-CIM Automation Suite can access during the processing of a job.

New Knowledge Drawing

To create a New Knowledge drawing, select the appropriate button under the 'Knowledge and Post Info' window.

Knowledge and Post Info Preview

Primary Knowledge: WOOD_SCIM_RS.dwg

Secondary Knowledge: WOOD_SCIM_RS.dwg

Use Primary Knowledge for Secondary Operations ☒

Primary Machine: RS274D.\$PP

Secondary Machine: RS274D.\$PP

Use Primary Machine for Secondary Operations ☒

Material: MATERIAL_0.5000

DOIT File: SCIM_WOOD_DOIT.dat

New Knowledge Drawing

New Drawing will allow you to have a new Knowledge drawing opened in AutoCAD. This will open the NewKnow.dwg drawing that is included with Router-CIM Automation Suite so that you can create cut knowledges or edit fixtures or table drawings in the knowledge drawing that you have selected. Router-CIM Automation Suite will load with the default 'ROUTER' configuration. At this time, make sure to select the correct Post Processor to match the cut knowledges that you will be creating when the 'Configuration Wizard' is opened.

For more information on the 'Configuration Wizard', click [here](#).

Edit a Knowledge Drawing

To edit a knowledge drawing, select the appropriate button under the 'Knowledge and Post Info' window.

Knowledge and Post Info Preview

Primary Knowledge: WOOD_SCIM_RS.dwg

Secondary Knowledge: WOOD_SCIM_RS.dwg

Use Primary Knowledge for Secondary Operations ☒

Primary Machine: RS274D.\$PP

Secondary Machine: RS274D.\$PP

Use Primary Machine for Secondary Operations ☒

Material: MATERIAL_0.5000

DOIT File: SCIM_WOOD_DOIT.dat

Edit Knowledge Drawing

Edit Drawing will allow you to have the Knowledge drawing opened in AutoCAD, with Router-CIM running so that you can add, change, or delete cut knowledges or edit fixtures or table drawings in the knowledge drawing that you have selected. Router-CIM will load with the same configuration that the knowledge drawing was made with originally. For instance, if the drawing was configured for a specific post processor when it was made, then it will load into Router-CIM with that post processor in use when selected for editing.

CAUTION: If you have more than one knowledge drawing and one post processor, you must select the correct knowledge drawing you want and the correct post processor to edit first before hitting the 'Edit' button. If you select the incorrect knowledge drawing with the incorrect post processor and select edit, AutoCAD will open with the incorrect post processor selected. If you change the post processor during the Router-CIM configuration wizard, the knowledge drawing will be purged and your knowledges will be erased.

For more information on creating knowledges, please refer to the [Router-CIM Manual](#) and follow the steps for creating a Knowledge.

DOIT File

Router-CIM Automation Suite runs on the drawing Layer to Cut Knowledge association principle. That means that for each Layer in a drawing that has geometry that you want cut on the machine, a Cut Knowledge must exist. Further, the two of them must be associated together in the system to allow Router-CIM Automation Suite to know that the partnership exists. From this area you may select a DOIT file (that is a Layer/Knowledge Association file) from the list of available DOIT files in the \Router-CIM\Automation\Doitdir folder. You can edit the selected file and add/remove associations, and even create new DOIT files for use in Router-CIM Automation Suite.

New DOIT File

In order to set up your layer-to-knowledge associations within a DOIT file, you need to create a DOIT file. The **'New'** button should be used. Make sure that you save the DOIT file you created in the proper location so that Router-CIM Automation Suite will know where it is located. The location that Router-CIM Automation Suite goes to is defined in under File/Settings under the tab ['System Folders'](#).

The screenshot shows the 'Knowledge and Post Info' dialog box. It has two tabs: 'Knowledge and Post Info' and 'Preview'. The 'Knowledge and Post Info' tab is active. It contains several sections with dropdown menus and buttons:

- Primary Knowledge:** Dropdown menu showing 'WOOD_SCIM_RS.dwg'. Below it are 'New' and 'Edit' buttons.
- Secondary Knowledge:** Dropdown menu showing 'WOOD_SCIM_RS.dwg'. Below it is an 'Edit' button.
- ☒ Use Primary Knowledge for Secondary Operations
- Primary Machine:** Dropdown menu showing 'RS274D.\$PP'. Below it are 'New' and 'Edit' buttons.
- Secondary Machine:** Dropdown menu showing 'RS274D.\$PP'. Below it is an 'Edit' button.
- ☒ Use Primary Machine for Secondary Operations
- Material:** Dropdown menu showing 'MATERIAL_0.5000'. Below it are 'New' and 'Edit' buttons.
- DOIT File:** Dropdown menu showing 'SCIM_WOOD_DOIT.dat'. Below it are 'New' and 'Edit' buttons, which are highlighted with a red rectangle.

Edit DOIT File

Once a DOIT file is created or you need to make changes to an existing DOIT file, you need to select the **'Edit'** button.

The screenshot shows the 'Knowledge and Post Info' dialog box with the following fields and buttons:

- Primary Knowledge:** WOOD_SCIM_RS.dwg (New, Edit buttons)
- Secondary Knowledge:** WOOD_SCIM_RS.dwg (Edit button)
- ☒ Use Primary Knowledge for Secondary Operations
- Primary Machine:** RS274D.\$PP (New, Edit buttons)
- Secondary Machine:** RS274D.\$PP (Edit button)
- ☒ Use Primary Machine for Secondary Operations
- Material:** MATERIAL_0.5000 (New, Edit buttons)
- DOIT File:** SCIM_WOOD_DOIT.dat (New, Edit buttons - highlighted with a red box)

Once you select the **'Edit'** button, the DOIT file editor will open.

The screenshot shows the 'Knowledge to Layer Association' dialog box with the following sections:

- Layer to Part Viewer:** A list of layers including 0, Cutout_0.5000, Cutout_10.0000_0.5000, Hole_0.1875_Thru, Hole_0.1969_Thru, Hole_0.5000_Thru, Outside_0.5000, PktCutout_0.4183_250, etc.
- Knowledge:** A list of knowledge items including CLC_VGR90_BLD_WD, COMMONLINE, DRL_1875_BLD_WD, DRL_1875_BLD_WD, DRL_1875_Thru, INS_2500_BLD_WD, INS_2500_THR_WD, INS_375C_THR_WD, INS_500C_THR_WD, INS_5000_BLD_WD, OTS_500C_ONS_WD, OTS_500C_THR_WD, PATREC_BLD_WD, PATREC_THR_WD, PKT_2500_BLD_WD, PKT_5000_BLD_WD, ROUND, SCRAP, SKELETON, STAYDOWN, etc.
- Layers:** A list of layers including 0, Cutout_0.5000, Cutout_10.0000_0.5000, Hole_0.1875_Thru, Hole_0.1969_Thru, Hole_0.5000_Thru, Outside_0.5000, PktCutout_0.4183_250, etc.
- Add:** A green circle with a plus sign.
- Move Association:** A blue circle with a plus sign.
- Knowledge to Layer Association:** A table with columns: TurnOff, Index, Knowledge, Layer. It lists various knowledge items and their associated layers.

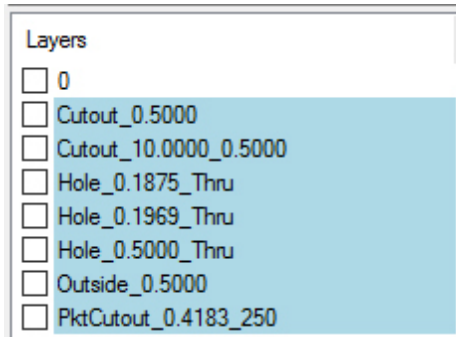
The DOIT file editor will search your currently selected knowledge drawing and the parts that included in the currently selected job. On the left hand side you will see three columns, **Layer to Part Viewer**, **Knowledge** and **Layers**. These columns represent the result of the search that was completed by Router-CIM Automation Suite so you will be able to see what knowledges are available to you, what layers are present in the job's drawings and which layers are on which parts.

The color assigned to a knowledge or layer represents one of the following. If the Layer is not a specified color, then no association to the layer exists and Router-CIM Automation Suite will not apply any cut knowledges to that layer.

Knowledge to Layer Association Color Definitions:

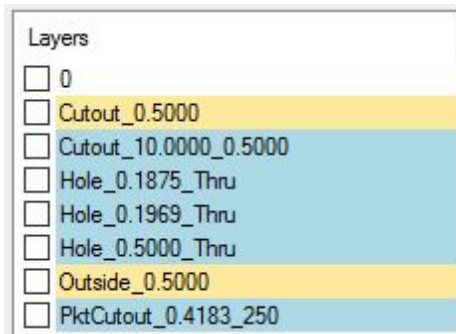
Layer Association Exists

When you see the layer found by the DOIT file in the **BLUE** color, this means that in the DOIT file itself, there is an association to a cut knowledge that IS present in the Knowledge drawing:



Layer Association Turned Off

When you see the layer found by the DOIT file in the **YELLOW** color, this means that in the DOIT file itself, there is an association to a cut knowledge that IS present BUT the association is Turned Off:



Knowledge Associated to Layer is Missing

When you see the layer found by the DOIT file in the **RED** color, this means that in the DOIT file itself, there is an association to a cut knowledge that IS NOT present:

Layers

- ☐ 0
- ☐ Cutout_0.5000
- ☐ Cutout_10.0000_0.5000
- ☐ Hole_0.1875_Thru
- ☐ Hole_0.1969_Thru
- ☐ Hole_0.5000_Thru
- ☐ Outside_0.5000
- ☐ PktCutout_0.4183_250

Knowledge to Layer Associations used in Current Job

In the DOIT file shown on the right side of the DOIT Editor, associations that are being used in the current job will be marked in bright **YELLOW**:

Knowledge to Layer Association

Frontside Backside

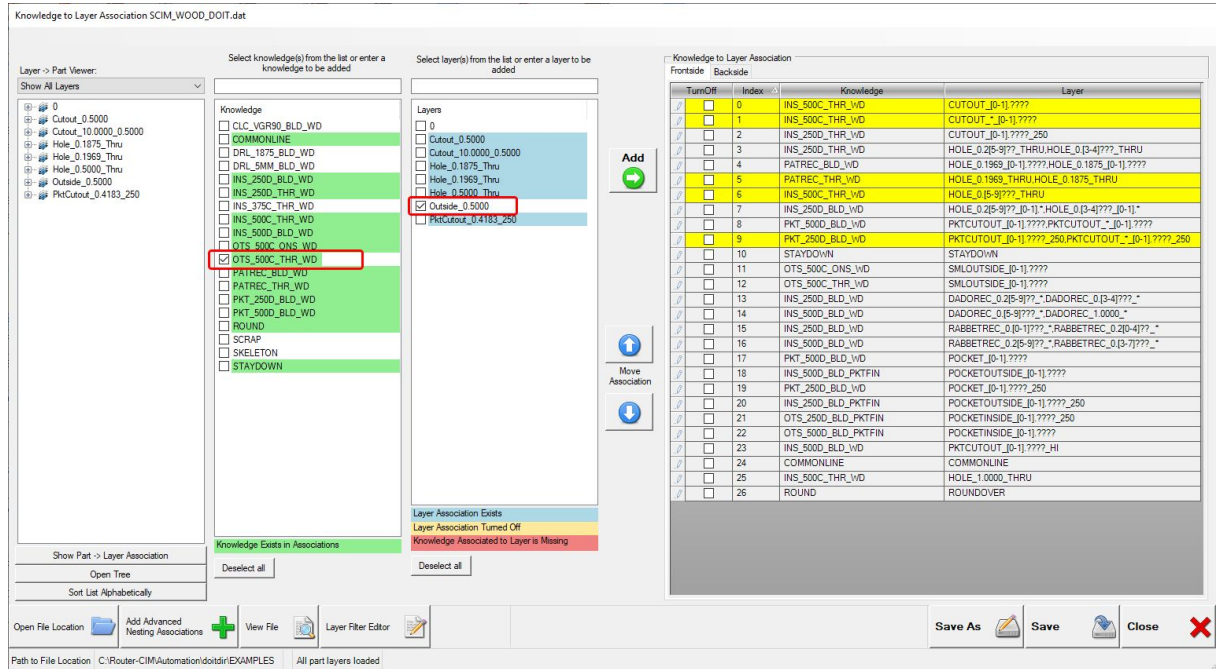
TurnOff	Index	Knowledge	Layer
<input type="checkbox"/>	19	PATREC_THR_WD	HOLE_0.1969_THRU,HOLE_0.1875_THRU
<input type="checkbox"/>	20	PKT_250D_BLD_WD	PKTCUTOUT_[0-1].????_250,PKTCUTOUT_*_[0-1].???
<input type="checkbox"/>	21	PKT_250D_BLD_WD	POCKET_[0-1].????_250
<input type="checkbox"/>	22	PKT_500D_BLD_WD	PKTCUTOUT_[0-1].????,PKTCUTOUT_*_[0-1].???
<input type="checkbox"/>	23	PKT_500D_BLD_WD	POCKET_[0-1].????
<input type="checkbox"/>	24	ROUND	ROUND OVER
<input type="checkbox"/>	25	STAYDOWN	STAYDOWN
<input checked="" type="checkbox"/>	26	OTS_500C_THR_WD	OUTSIDE_0.5000

Creating a Knowledge to Layer Association

In the below example, you can see that layer 'OUTSIDE_0.5000' does not have a knowledge associated with it. In order to make an association, you will need to select the correct knowledge that will be used to machine the layer. If you do not have a knowledge created for this layer, please go to ['Edit Knowledge Drawing'](#) for information on how to add a knowledge.

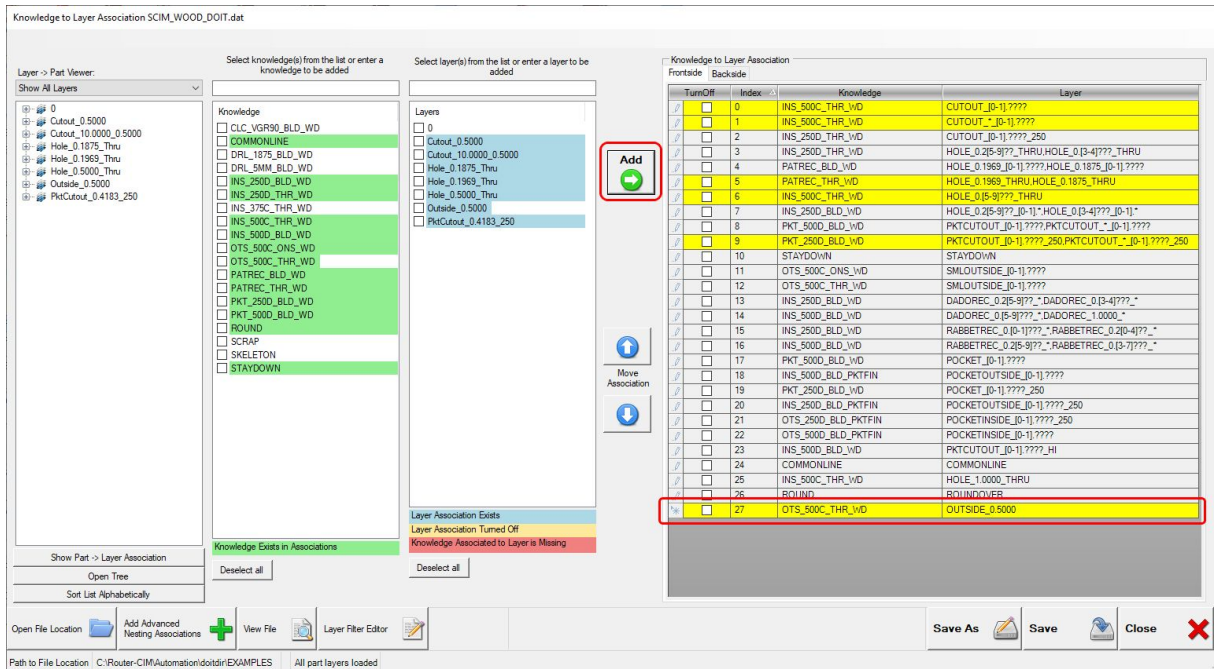
Note: The knowledge does not have to be named exactly the same as the layer.

To make the association, select the correct cut knowledge and the correct layer on the left hand side of the window by selecting the check box.



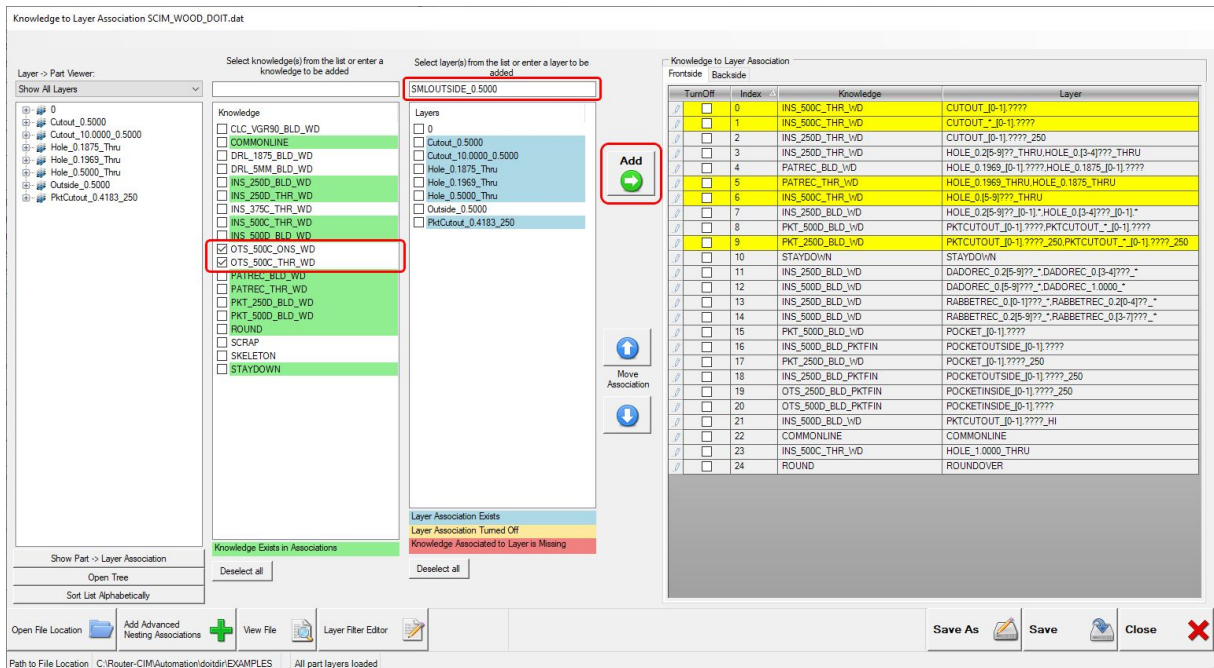
Note: You can select multiple knowledges to associate with one layer or one knowledge to associate with multiple layers. Layers and Knowledges can be used more than one time.

Once you have selected the correct knowledge and layer, select the 'Add' button.



Once you select **'Add'**, you can see that the knowledge "OTS_500C_THR_WD" is now associated with layer "OUTSIDE_0.5000". This will mean that every time Router-CIM Automation Suite sees the layer "OUTSIDE_0.5000" it will use the cutting parameters defined in the knowledge "OTS_500C_THR_WD".

You can also manually type in cut knowledge names and layer names. This is common for SML associations when using the 'Rename Outside Layers' feature of Router-CIM Automation Suite.



When a cut knowledge is associated to a layer, the DOIT file editor will update the color definition automatically.

Creating a Knowledge to Layer Association using Wildcards

You can also use wildcards in order to make your Knowledge to Layer associations. The wildcard is a powerful tool that allows Router-CIM Automation Suite to use a knowledge on a layer that does not match exactly the layer name.

NOTE: Wildcards are a very powerful tool. They should be used with caution so that you do not have Router-CIM Automation Suite using cut knowledges for layers that you do not want to cut in that particular way.

Wildcards come in multiple different forms. Here is a list of acceptable options:

Wildcard	Description
*	The * (asterisk) is a way to ignore/discard any information that comes after it. An example would be if you were trying to associate the same knowledge to a layer called OUTSIDE_0.2500, OUTSIDE_0.5000 and OUTSIDE_0.7500, you could simply type in the layer text box OUTSIDE_0.* and associate this to the correct knowledge. As long as OUTSIDE_0. is at the beginning of the layer name, the rest of the layer name will be ignored.
Multiple *	<p>You can also use multiple asterisks in order to filter out portions of a layer name. An example would be if you wanted to drill all holes that were any diameter from 0.2 to 0.29 and 0.5 inches deep with a 0.25 drill bit. If your layers included the diameter and depth, you may end up with many layers depending on how the circle was drawn. You can use multiple asterisks to accommodate this situation like HOLE_0.2*_0.5*. Associating your 0.25 drill knowledge to this layer would cover all circles drawn from 0.2 to 0.2999 and had depths of 0.5 to 0.5999.</p> <p>To summarize, if multiple asterisks are used, the association will ignore what is after the first asterisk but then will need to see the continuation of the layer name if there is something present after it.</p>
?	The ? (question mark) is defined as a single character. The difference here is that it does not ignore what's after it but only ignores a single character where the ? is found. An example would be if you are using a single digit as a trigger for something to happen. If I have a layer PKTCUTOUT_0.2501 where the final "1" represents picking a different tool. I could associate the correct knowledge to a layer like PKTCUTOUT_0.???1. This would mean that any pocket depth as long as there are 3 characters after the 0. and the fourth character would need to be a "1" in order for this association to happen.
[]	<p>The [] will allow you to specify a range for a single character. An example would be to associate one knowledge for layers that range from DADOREC_0.2000_0.5000 to DADOREC_0.4999_0.2500 where you were not concerned about the depth.</p> <p>The knowledge would be associated to a layer like this, DADOREC_0.[2-4][0-9][0-9][0-9]_*.</p>
,	The , (comma) will allow you to associate one knowledge to multiple layers in one line of the DOIT file.

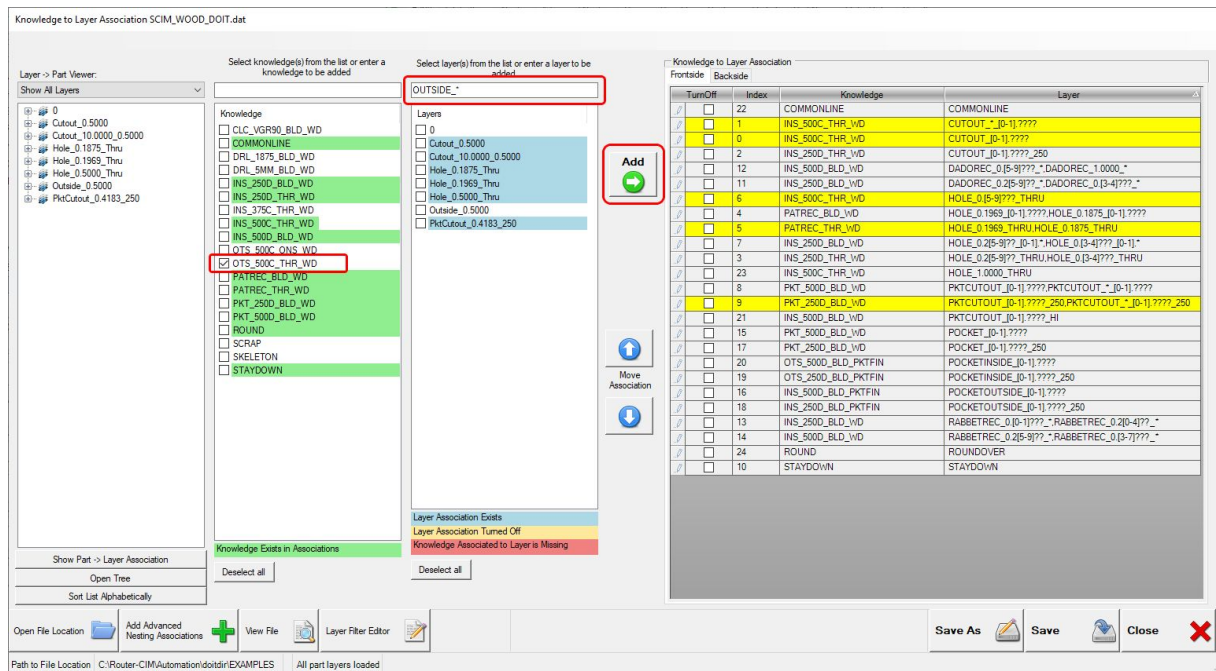
For example if I want the knowledge of OUTER_PROFILE to be associated to OUTSIDE_0.2500, OUTSIDE_0.5000 and OUTSIDE_0.7500, you would select the correct knowledge and type in to the text box above layers
OUTSIDE_0.5000,OUTSIDE_0.7500,OUTSIDE_0.2500 and select the ADD button. This one association will cover all three of the layers.

Here is an example DOIT file that shows some of the wildcard options:

Knowledge to Layer Association			
Front Side		Back Side	
Turn Off	Index	Knowledge	Layer
<input checked="" type="checkbox"/>	0	DADO_250	DADOREC_0.[2-4][0-9][0-9][0-9]_*
<input type="checkbox"/>	1	5MM_DRILLS	HOLE_0.1969_*
<input type="checkbox"/>	2	8MM_DRILLS	HOLE_0.3150_*
<input type="checkbox"/>	3	OUTSIDE_PROFILE	OUTSIDE_0.5000,OUTSIDE_0.7500,OUTSIDE_0.2500
<input type="checkbox"/>	4	DADO_250	PKTCUTOUT_0.???2
<input type="checkbox"/>	5	DADO_250	PKTCUTOUT_0.???3
<input type="checkbox"/>	6	DADO_250	RABBETREC_0.[2-4][0-9][0-9][0-9]_*
<input type="checkbox"/>	7	OUTSIDE_PROFILE	SML*_0.*
<input type="checkbox"/>	8	SMLOUT_PROFILE	SML*_0.*

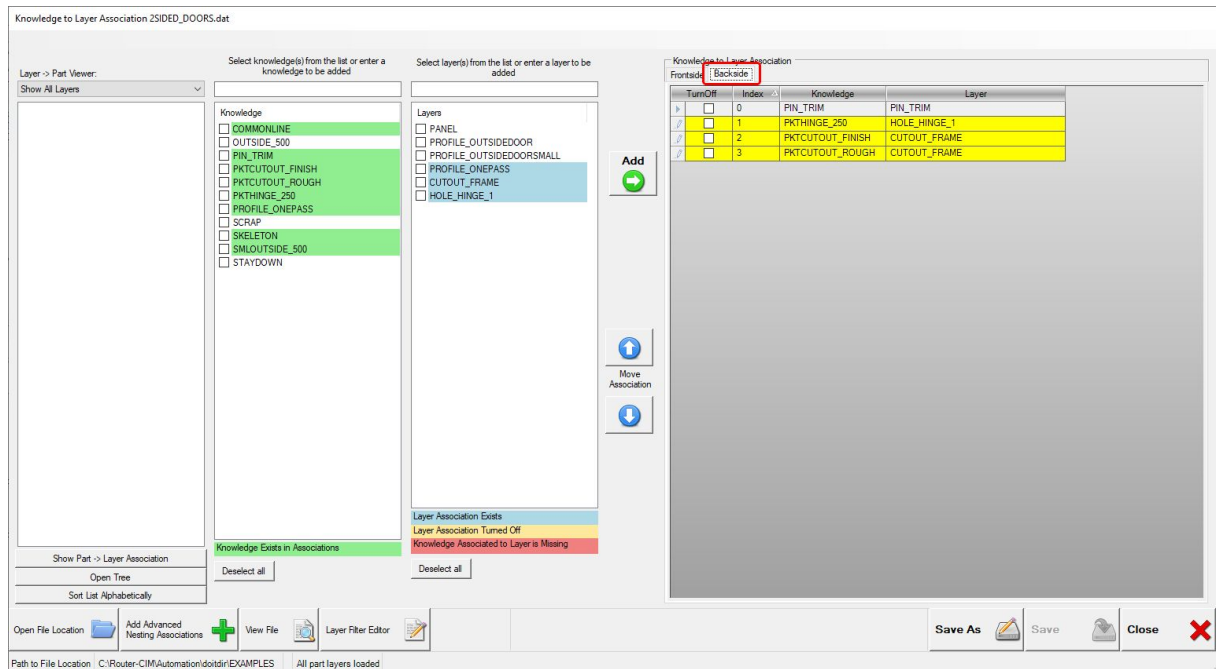
To add a Wildcard association, please follow this procedure:

If I wanted to associate the knowledge "OUTSIDE_375" to all layers that begin with "OUTSIDE_", you would set up the association to as follows:

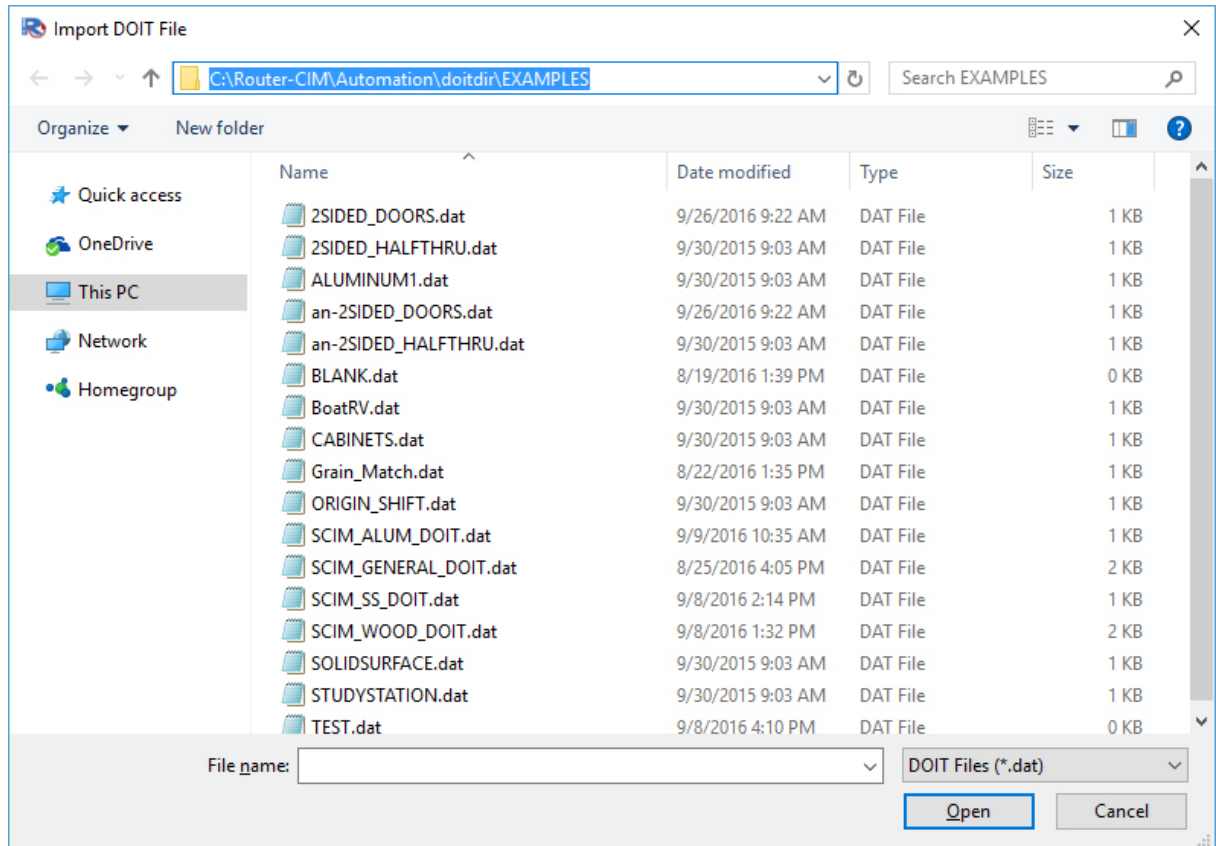


Adding Backside Associations

In order to use the advanced nesting module component "**Enable Backside Nesting**", you will need to complete the same process for layers that you want to machine on the backside of the part by selecting the "Backside" tab on the right side and repeating the process for all the backside operations. See ['Automated Two-Side Nesting'](#) for more information.



By selecting the '**Open File Location**', it will open a windows browser showing you where your DOIT file is located. The file type will be a .DAT extension.



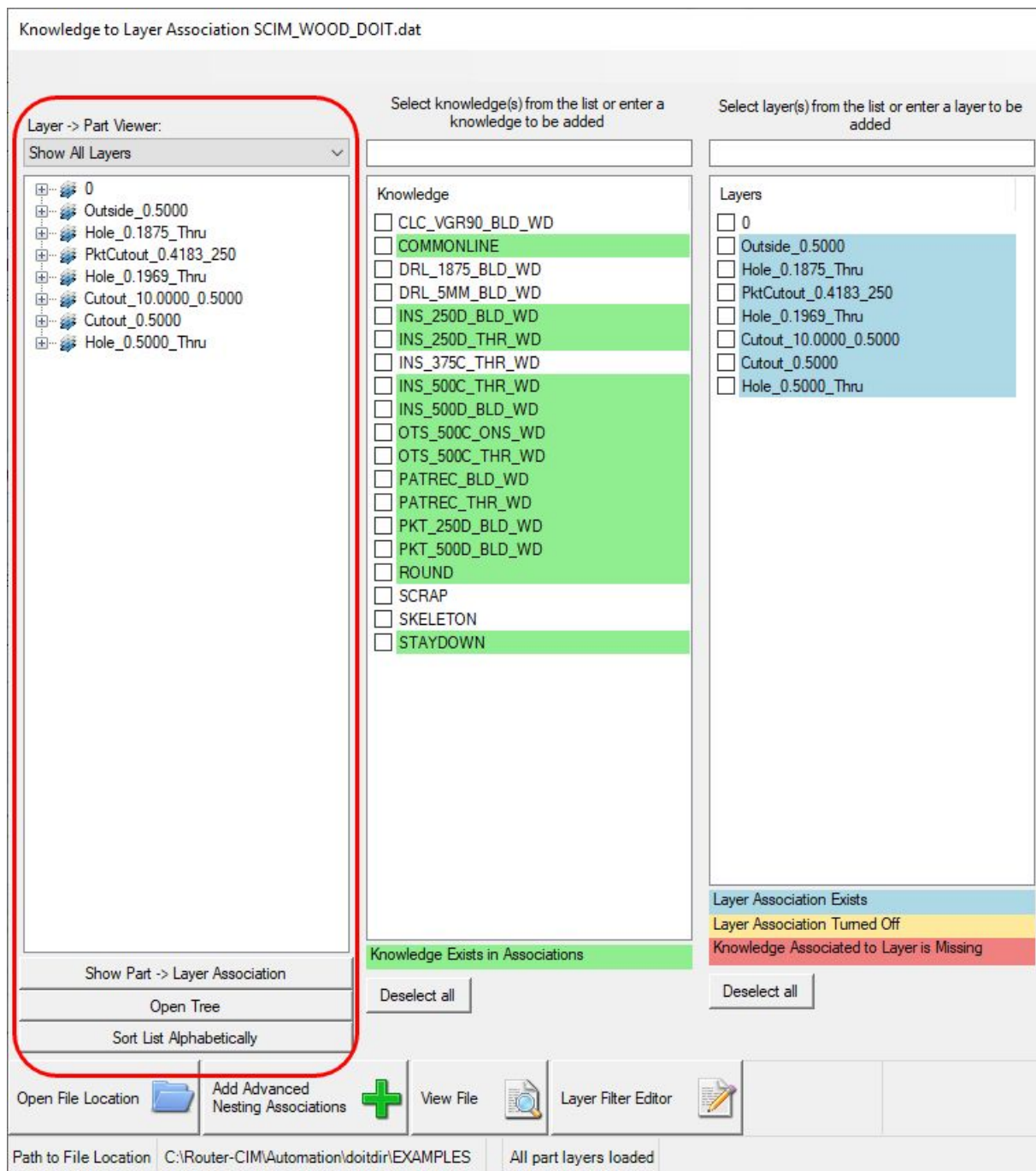
By selecting the **'Add Advanced Nesting Associations'** button, Router-CIM Automation Suite will search your knowledges to find the STAYDOWN, COMMONLINE, SKELETON and PIN_TRIM knowledges and automatically create the layer-to-knowledge association. This button also works if your STAYDOWN, COMMONLINE, SKELETON and PIN_TRIM knowledges begin with the Material Code.

After you have made the changes to the DOIT file, you can **'Save'** the file or use the **'Save As'** in order to give it a new name. The file must be saved in order to have Router-CIM Automation Suite use the layer-to-knowledge associations that you have defined.

Once the file is saved, you can select the **'Close'** button.

Using the Layer to Part/Part to Layer Viewer

The Layer to Part/Part to Layer Viewer allows you to see either the Layers that the DOIT file found or the parts that are in the current job when the DOIT file was opened.



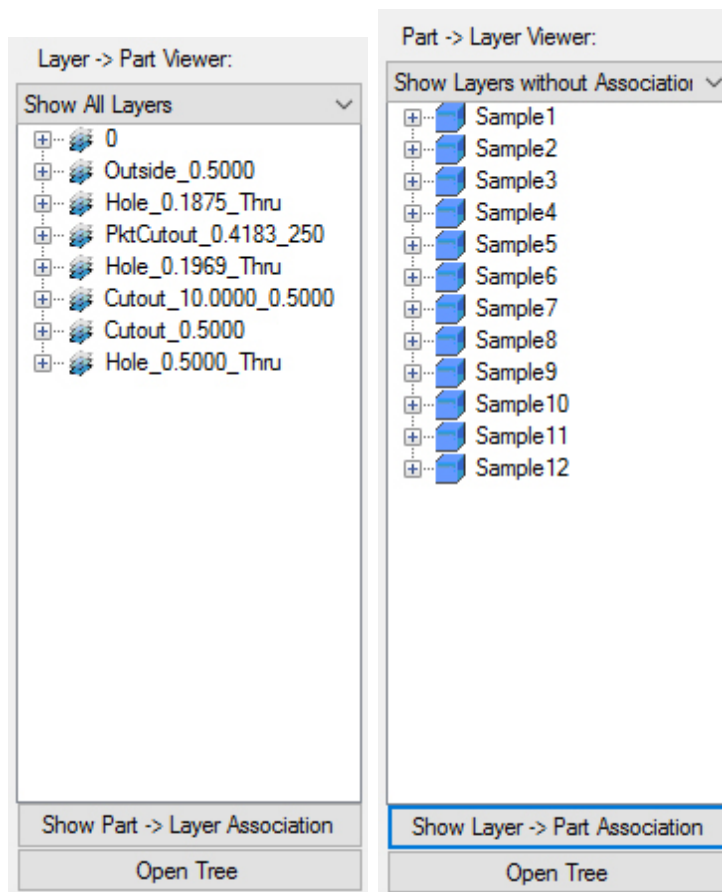
Show Layer to Part: This will show which parts the DOIT Editor found using the selected layer

Show Part to Layer: This will show which layers the DOIT Editor found in that particular part

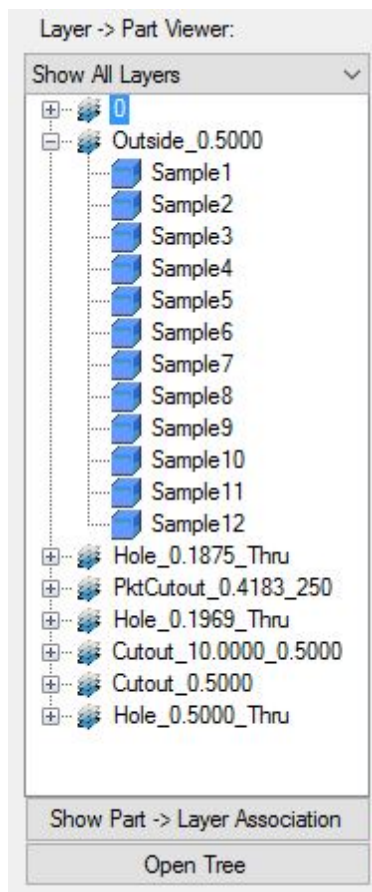
You can switch between the two options by selecting the Show Part -> Layer Association or Show Layer -> Part Association.

Layer to Part Association:

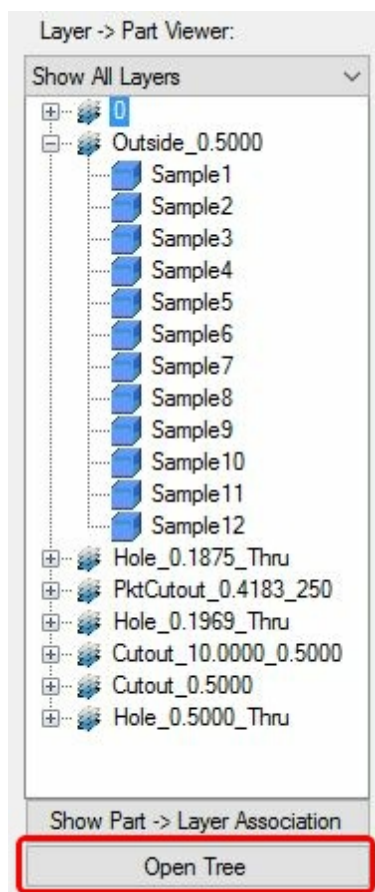
Part to Layer Association:



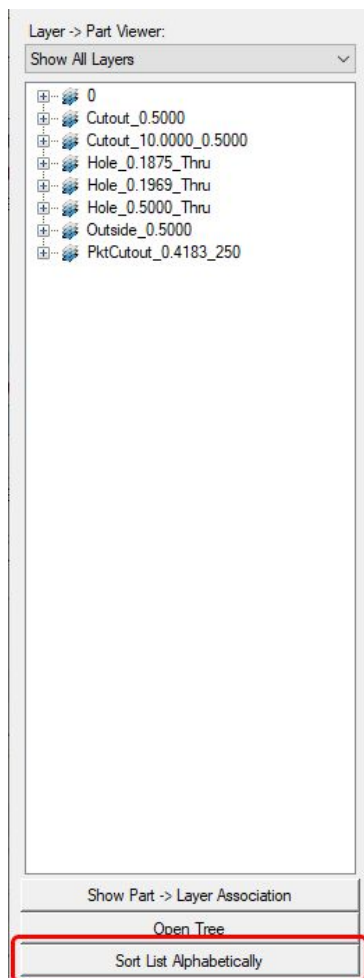
Each line will have an expand option shown as the '+' symbol. When selected, the selected line will expand showing you more details:



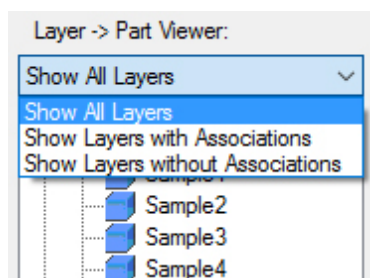
The '**Open Tree**' button will expand all the '+' symbols with one click:



The '**Sort List Alphabetically**' button will sort the layers for simple viewing:



You can decide what you want to see in the Layer to Part Viewer by using one of the sorting options:



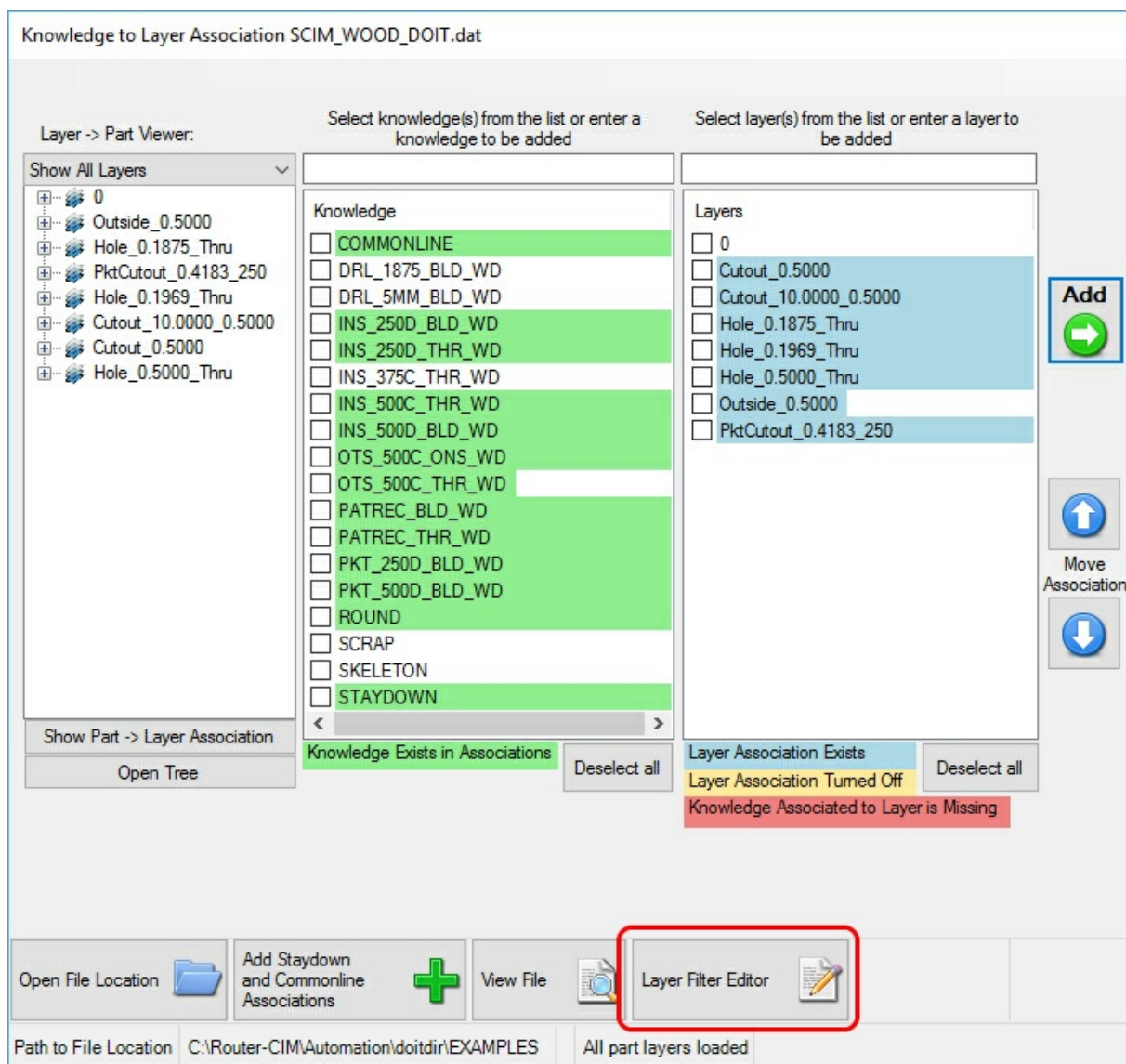
Show All Layers: Shows all the layers that the DOIT Editor found

Show Layers with Associations: Shows all the layers that the DOIT Editor found that had cut knowledges assigned

Show Layers without Associations: Shows all the layers that the DOIT Editor found that did not have cut knowledges assigned

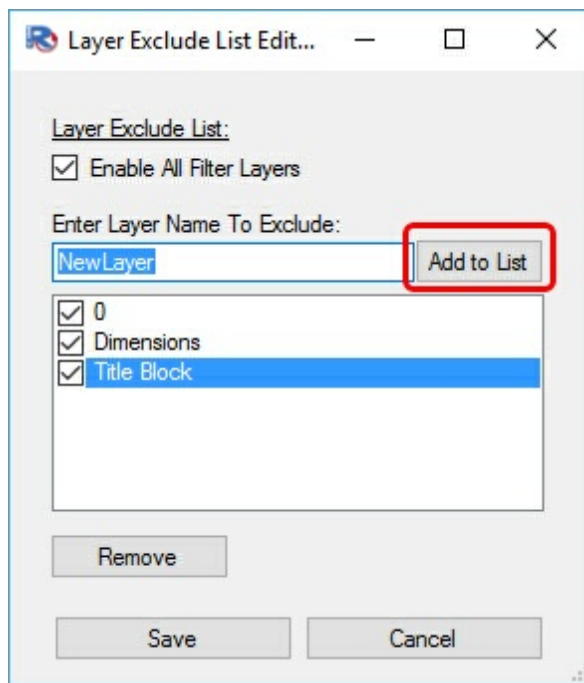
Using the Layer Filter Editor:

The Layer Filter Editor allows you to filter some of the layers that appear in the Layer to Part Viewer. This is common for AutoCAD default layers or layers that you may use in your drawings that are not relevant to Router-CIM Automation Suite. These layers may include things such as Dimension layers or Title Block layers, etc. The Layer Filter Editor allows you to only see the layers that you are interested in applying cut knowledges to.

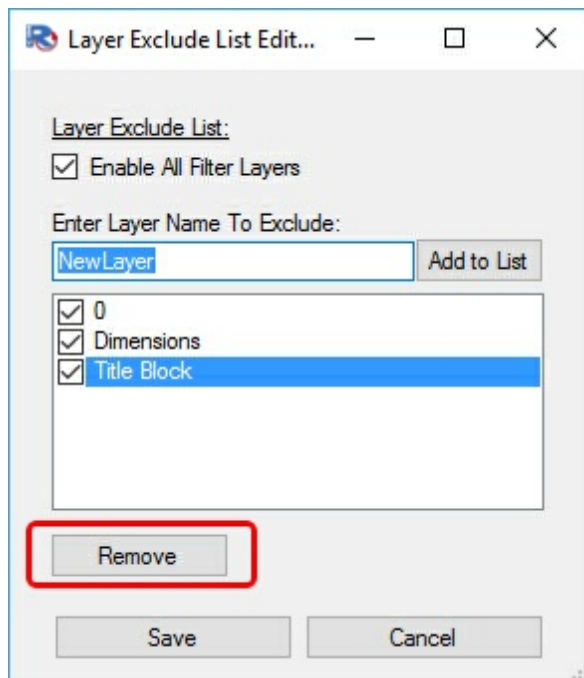


To Add A Layer to the Layer Filter Editor:

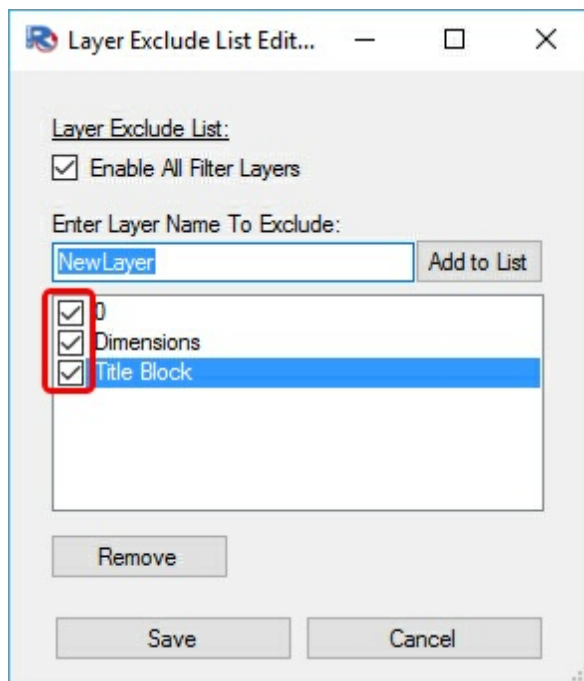
Simply type in the full layer name in the 'Enter Layer Name to Exclude' area and select the 'Add to List' button.

**To Remove A Layer from the Layer Filter Editor:**

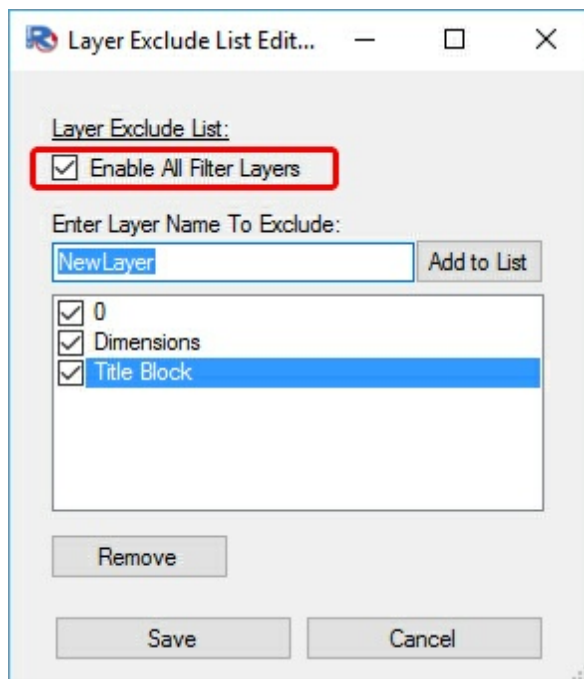
Simply highlight the layer you want to remove and select the 'Remove' button.



Once your Layer Filter list has been completed, you will need to check the box by the layers that you want the Layer Filter Editor to exclude:



If you want to select all the layers, simply check the box for 'Enable All Filter Layers':



Multi Post Processing

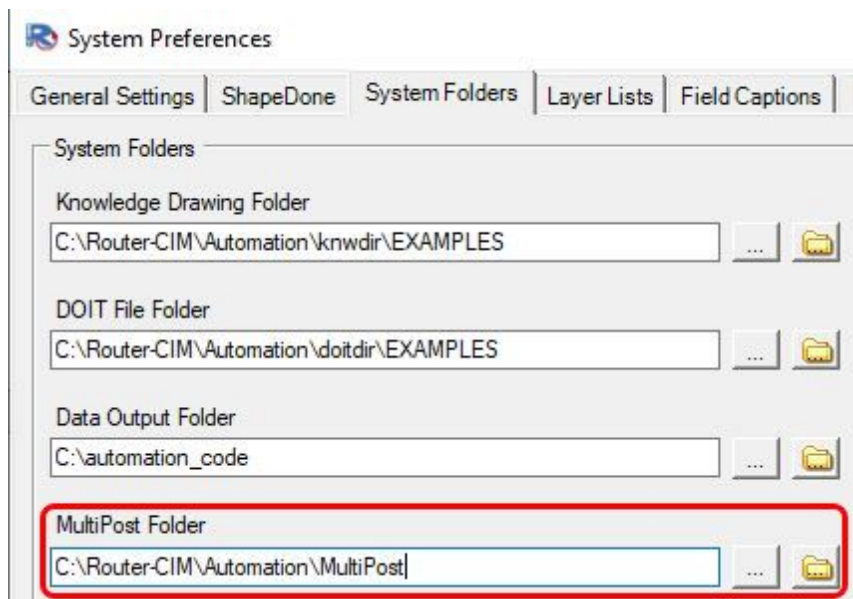
To setup a MultiPost process, follow these instructions.

Setting up MultiPost

In order to use MultiPost, you will need to create Pack and Go's of the job setups. The Pack and Go files will include items such as the Post Processor, knowledge drawing, DOIT file, System Preferences, Job Settings, Materials, etc.

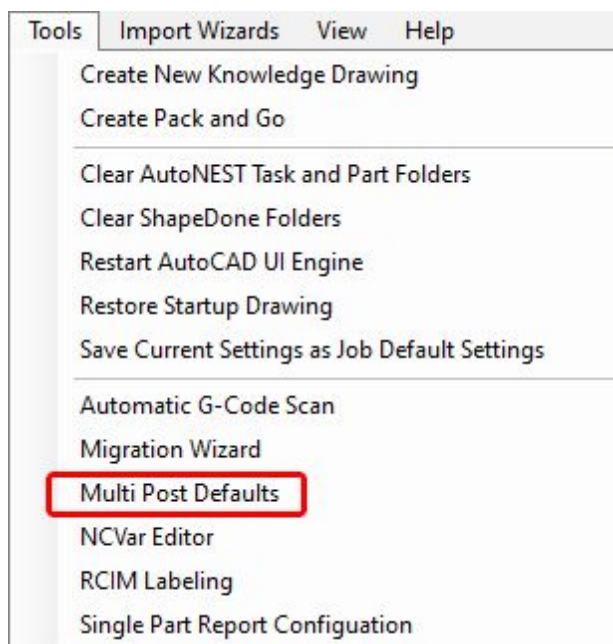
To create MultiPost defaults, you will need to create Pack and Go's (PAG) of the jobs that will represent each run through MultiPost. For information on how to create a PAG, refer to the ['Create Pack and Go'](#) section under the **'Tools'** section. The PAG that are created for use with MultiPost need to be stored in a location that Router-CIM Automation Suite can access. By default, the folder location is set in Router-CIM Automation Suite as C:\Router-CIM\Automation\Multipost. The location of this folder can be changed and should be changed if you are sharing the database to allow for multiple users to access the PAG files. You can change this folder location by adjusting the setting located in the System Preferences section located under ['System Folders'](#).

Note: You may have to create multiple databases if you want MultiPost processing to have access to different setups of materials such as a CNC with X axis as the long axis and another CNC you may be programming for having the Y axis as the long axis. Each PAG would need to be created in the corresponding material database.

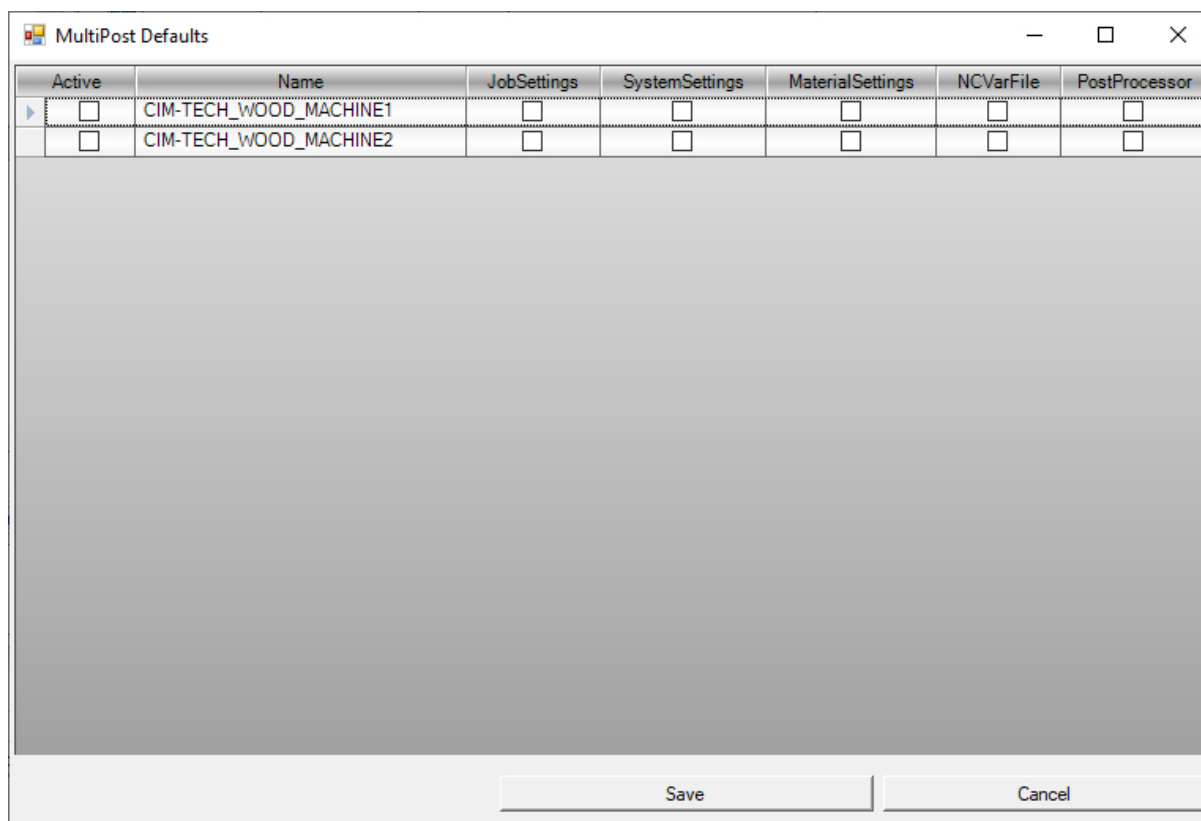


Once the PAG files have been created and saved to the correct folder location that you have set Router-CIM Automation Suite to, you are able to setup the MultiPost defaults.

To access the MultiPost defaults, go to the **'Tools'** section in the Menu Bar and select **'Multi Post Defaults'**.



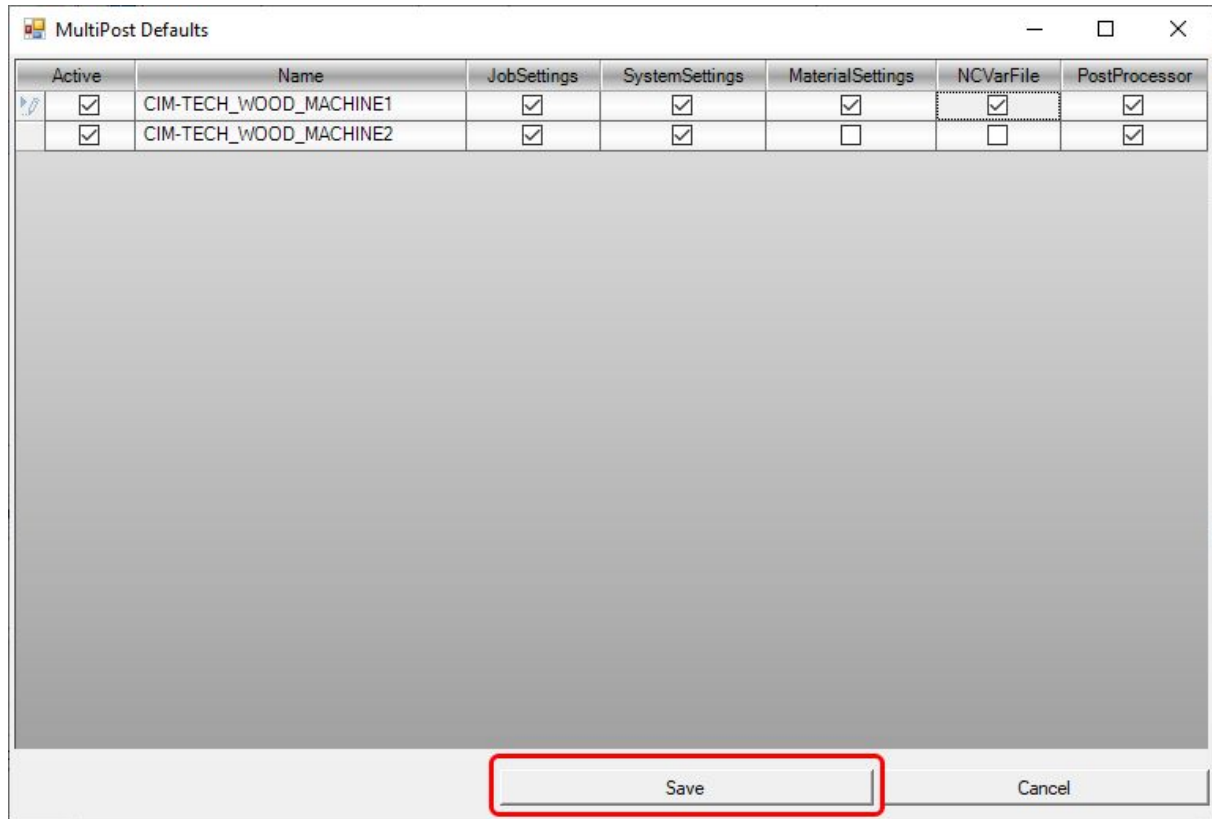
The MultiPost Defaults window will open. When the window opens, you should see a row for each of the PAG files that are located in the MultiPost default folder location.



With the window open, you can select the items that you want to be used when processing a MultiPost job. The user has the option of turning on and off the import of the job settings, system settings, material

settings, NCVar file, and the post processor. If they are not selected, the selected job settings will be used.

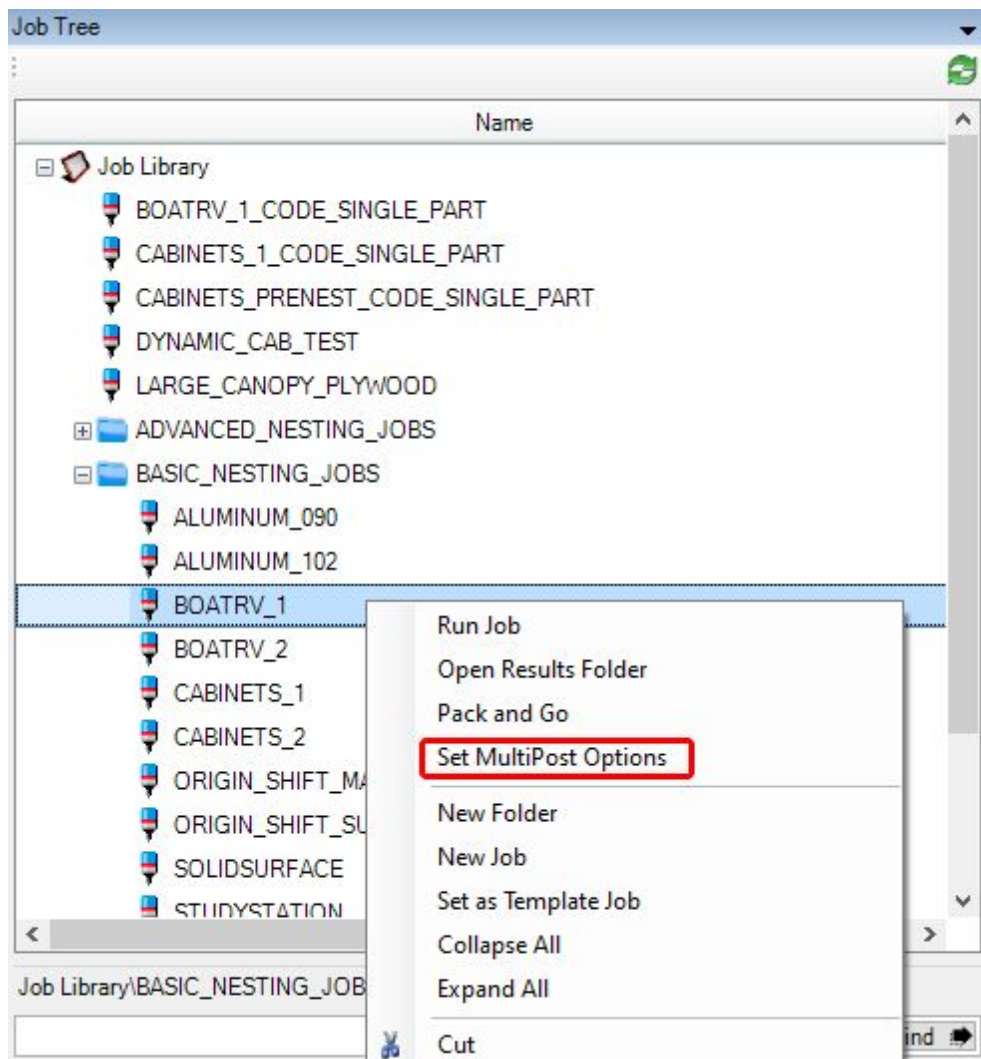
Once you have the settings changed to match the way that you want MultiPost to process jobs, select the 'Save' button.



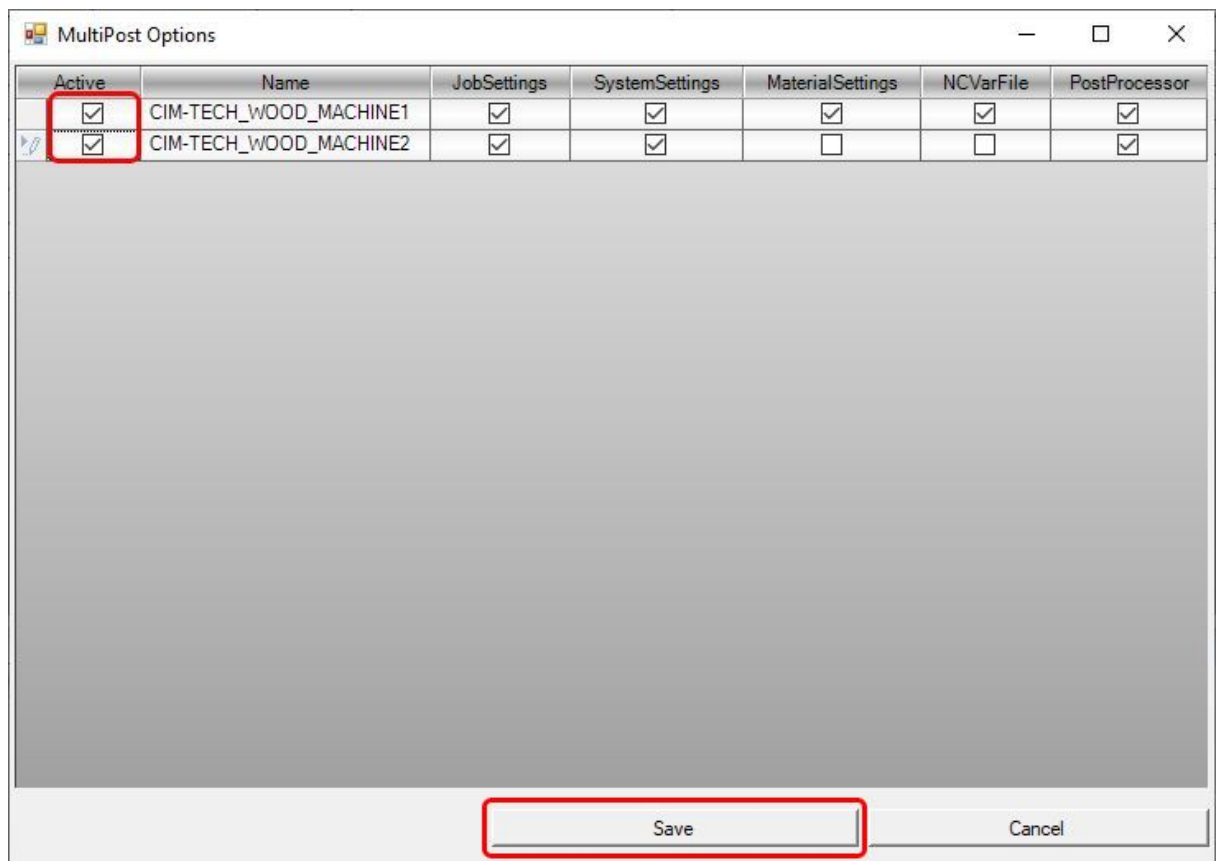
With the MultiPost defaults set, you are now ready to setup your MultiPost job in Router-CIM Automation Suite.

Creating a MultiPost Job

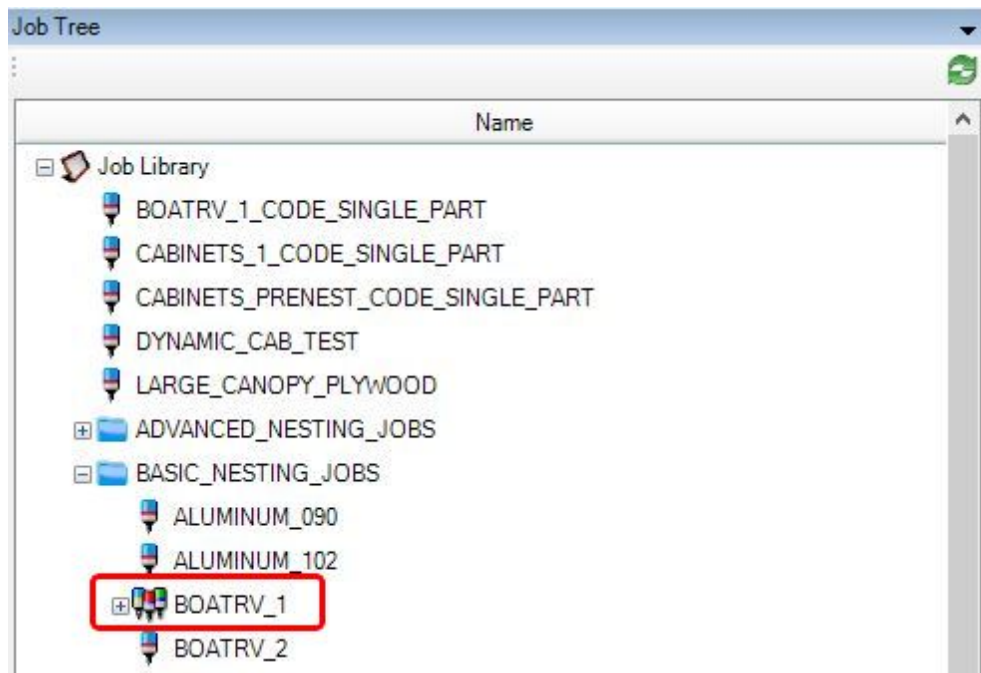
To create a job that uses the MultiPost processing, you will need to right-click on the job and select 'Set MultiPost Options'.



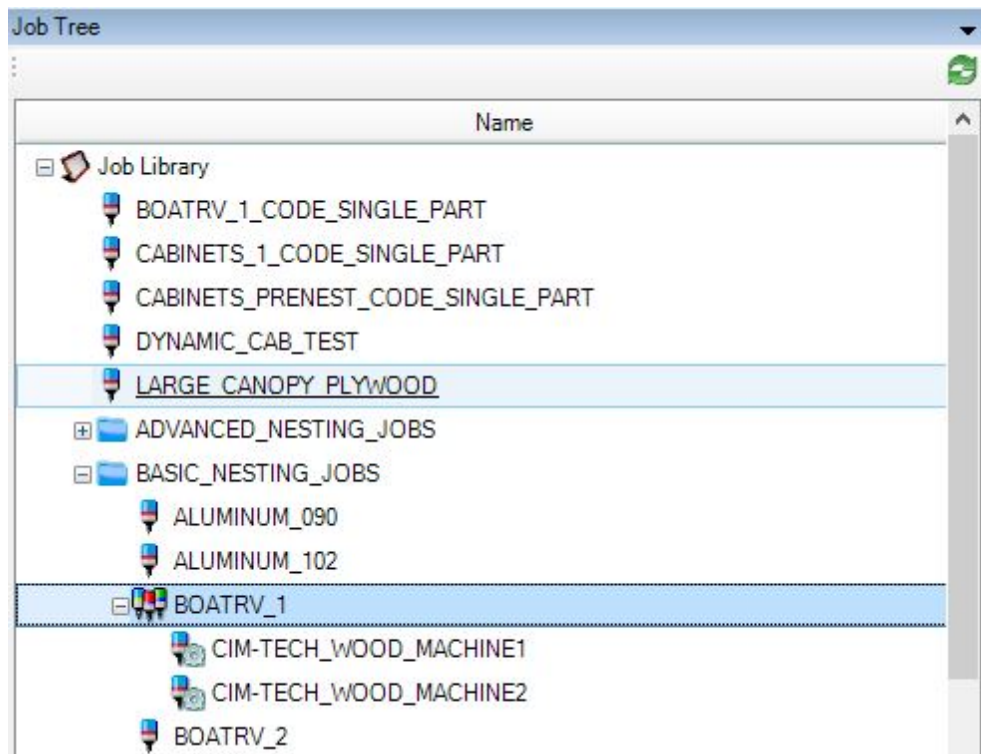
When selected, it will open the MultiPost Options window. From this window you will check the Active box next to the MultiPost defaults that you setup with the PAG files. If you have more MultiPost defaults setup and you do not need them for this MultiPost Job, simply do not make them 'Active' for this MultiPost job. Select the 'Save' button when you have activated the MultiPost defaults that are needed.



When a job has been formatted for MultiPost, you will not see a new icon next to the job.



If you select the plus sign next to the job, you can see the MultiPost defaults that were selected.



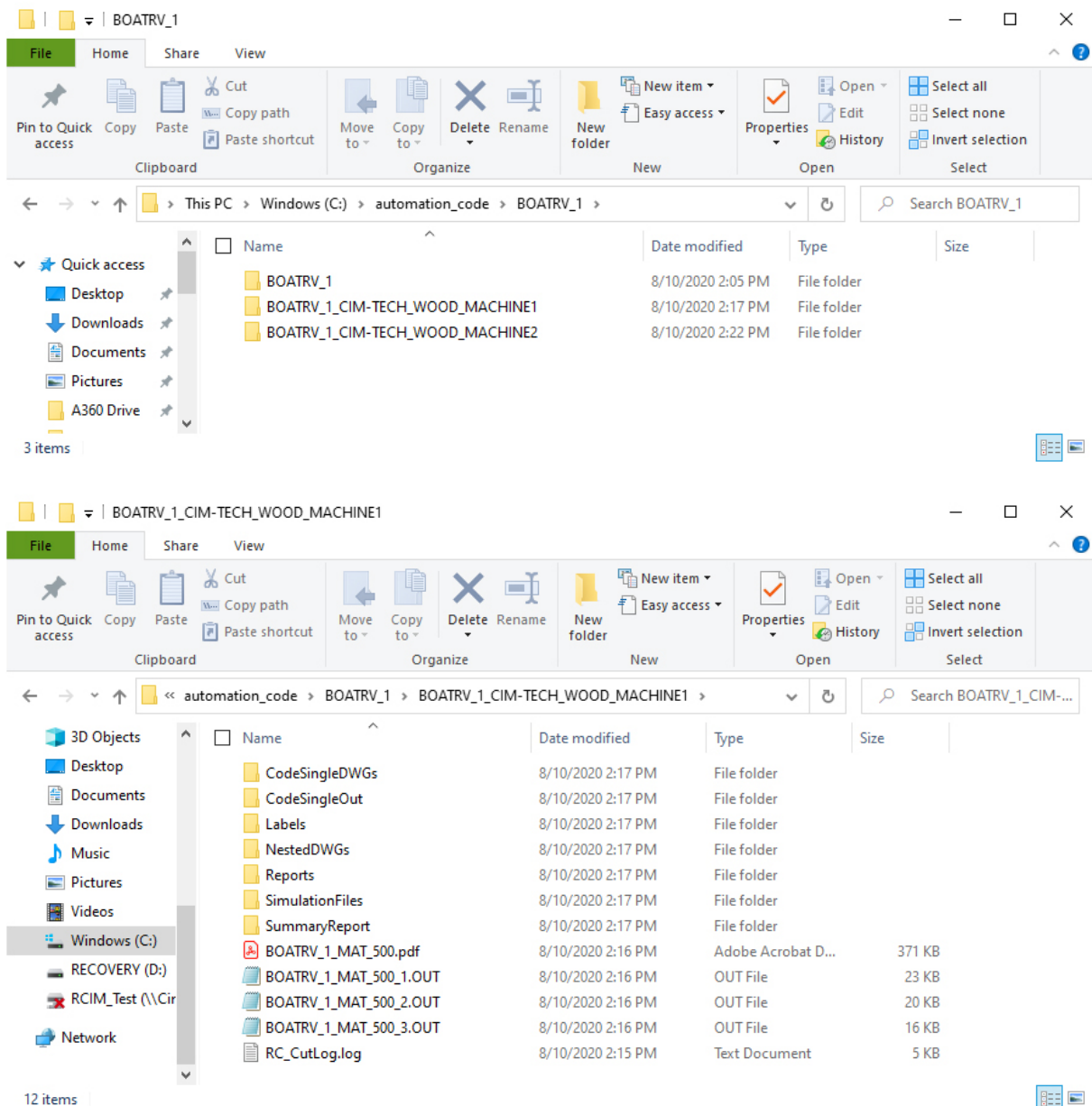
If you select the plus sign next to the job, you can see the MultiPost defaults that were selected.

To change a Router-CIM Automation Suite job back to a standard job, right-click on the MultiPost job and uncheck the active buttons on any MultiPost defaults selected and save.

Once a MultiPost job has been created, you simply need to select the 'Run Job' button to process the job.

MultiPost Job Results Folder

When the MultiPost job has completed, it is time to examine the 'Output Folder' contents. You will see multiple folders. One folder will be the original jobs result and then you will see additional folders that will have the results from each of the selected MultiPost options. Each folder will contain the relevant results and information based on your System Preferences settings.



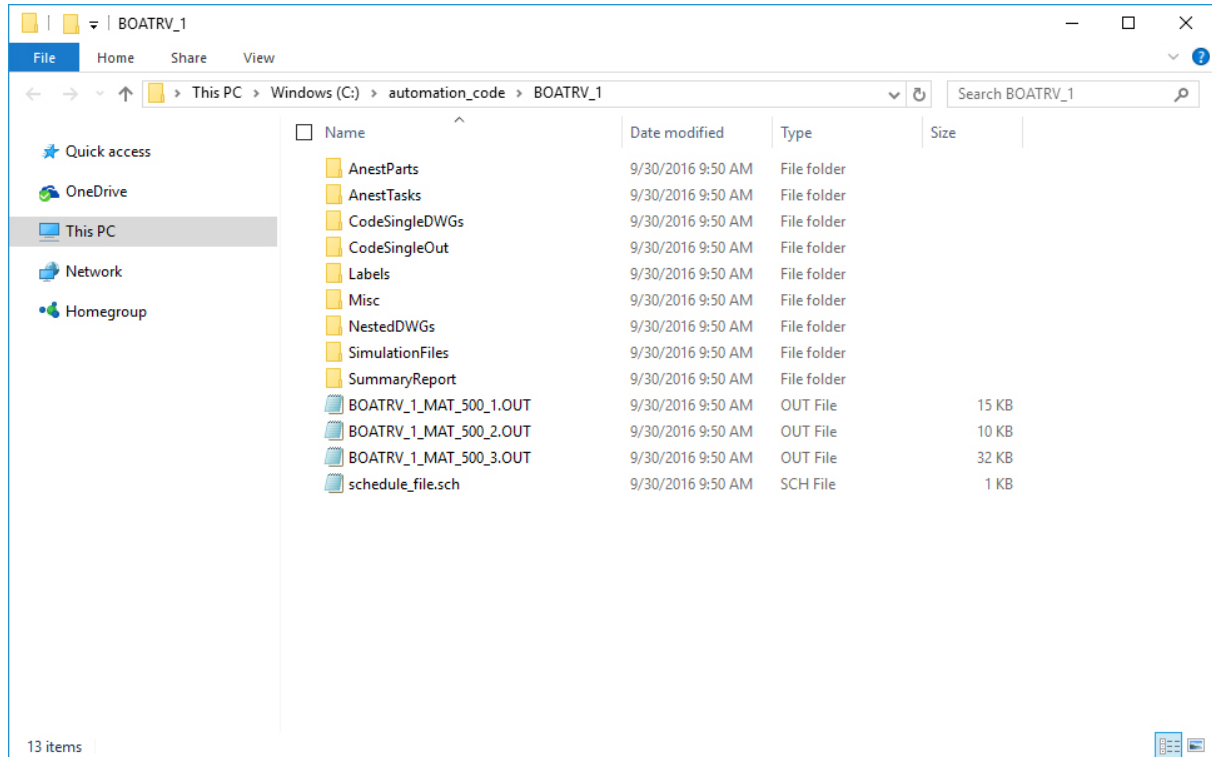
The Results Folder

After every job is run in Automation, there is a folder created with the same name as the job name. This folder contains all the files created by the job as a result of the run in Automation.

The result folder will have your NC Code files, summary reports, nested drawings, AutoNEST parts and tasks, schedule files, etc. The root folder (with the job name) will contain the NC code files, and schedule file for the job and also the separate folders for the other data created by the job.

The items in the will vary depending on the selections made located in File/Settings under the '[Systems Folder](#)' tab.

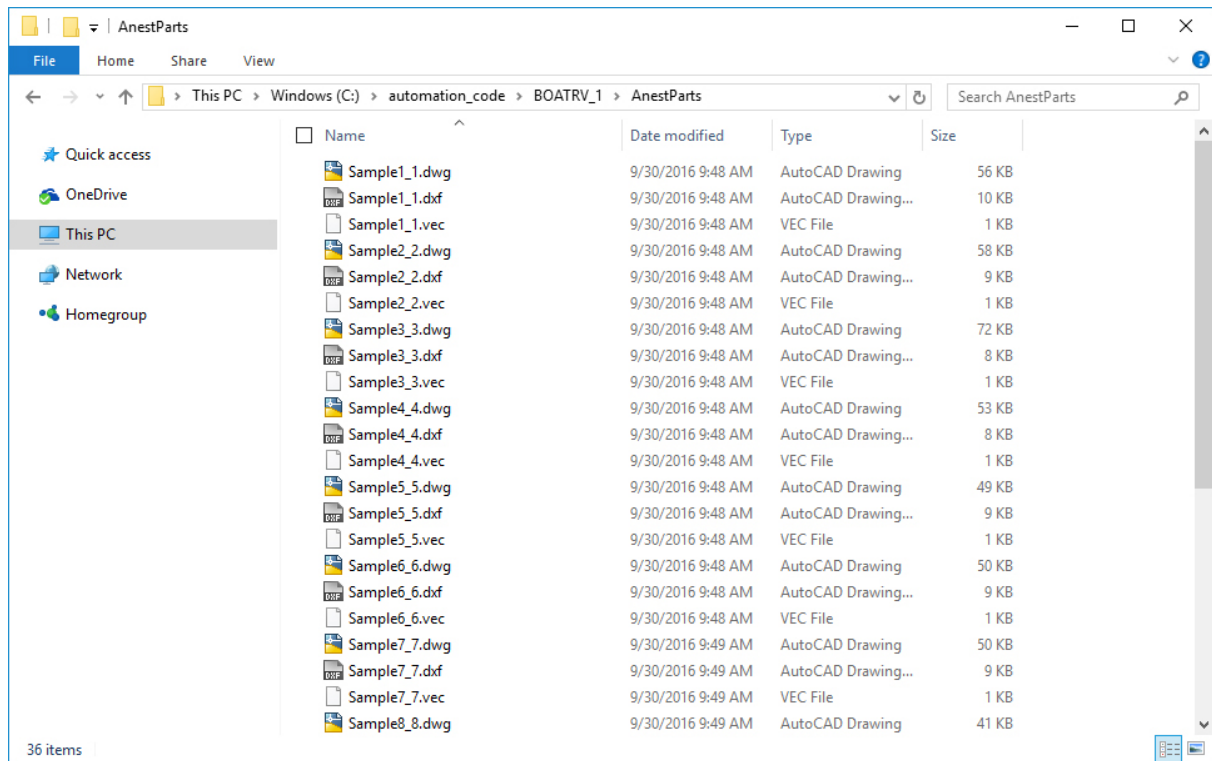
The location of the folder by default will be under C:/automation_code. The output location can be changed by changing the '[Data Output Folder](#)' setting located at File/Settings under the 'Systems Folder' tab.



AnestParts Subfolder

All of the part files (.dwg, .dxf & .vec) for each part will be stored in this folder for AutoNEST to recreate the parts with.

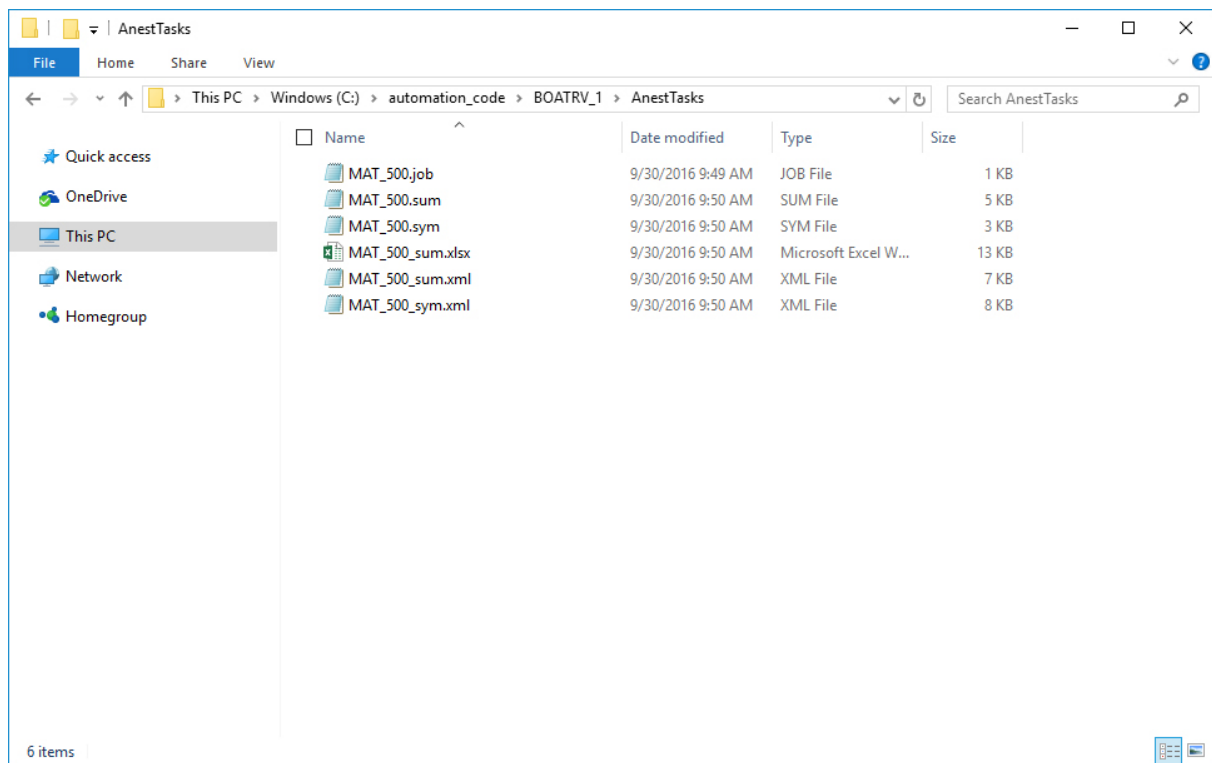
The folder will display only if you the selection was made to show this folder located in File/Settings under the '[Systems Folder](#)' tab.



AnestTasks Subfolder

All the task files (.sym, .sum, .sgd and .job) for the job that AutoNEST needs to recreate the job settings.

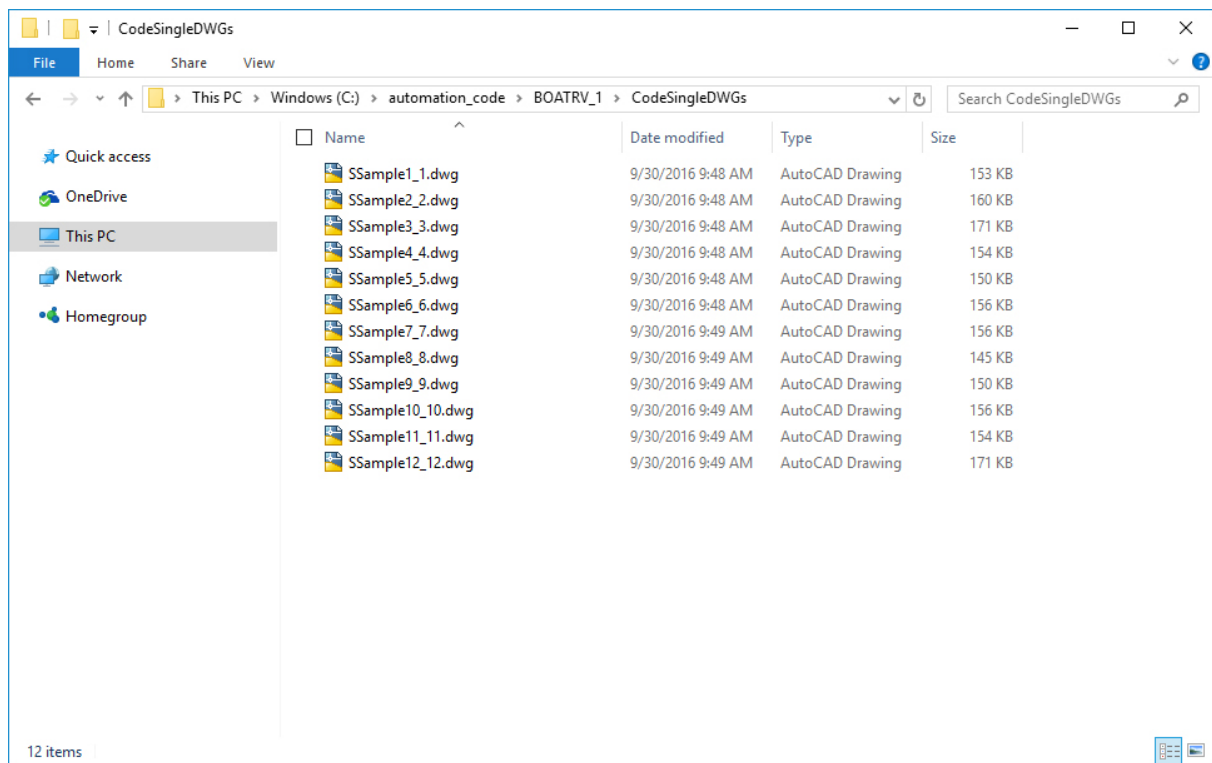
The folder will display only if you the selection was made to show this folder located in File/Settings under the '[Systems Folder](#)' tab.



Code Single DWGs

This folder contains the 'Code as Single Part' drawings for all the parts in the job. Each part will have a separate drawing containing all the cuts for that part.

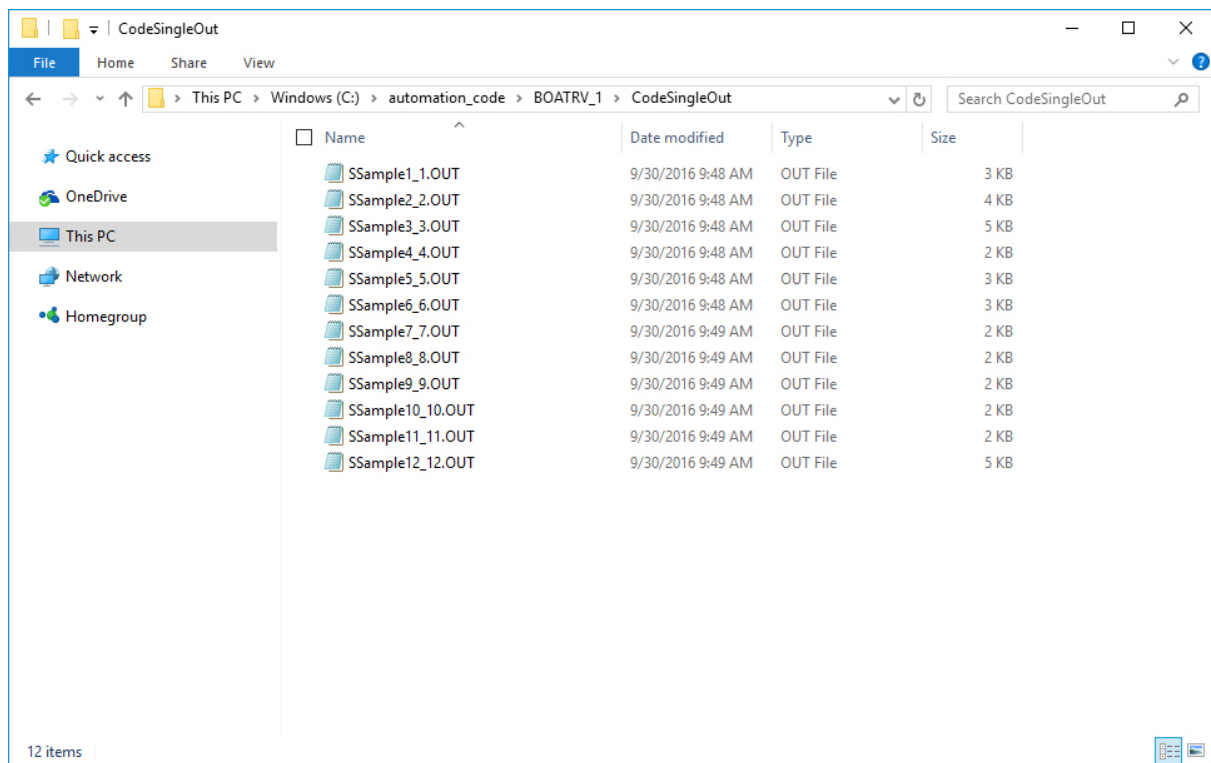
The folder will display only if you the selection was made to show this folder located in File/Settings under the '[Systems Folder](#)' tab.



Code Single Out

This folder contains the 'Code as Single Part' NC code files for all the parts in the job. Each part will have a separate NC code file containing all the code needed to process that part.

The folder will display only if you the selection was made to show this folder located in File/Settings under the '[Systems Folder](#)' tab.

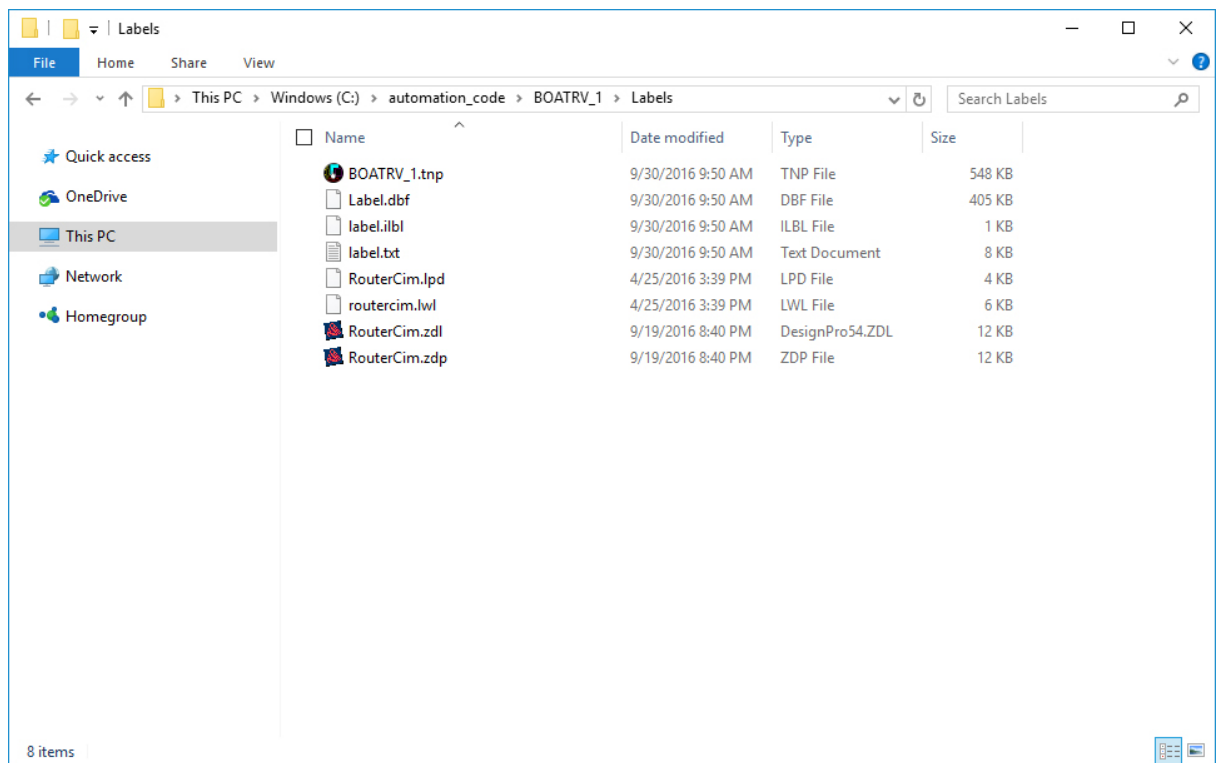


Labels Subfolder

All the label files for the job are stored in this folder including additional folders if using the RCIM Labeling. There are label files for Avery Design Pro, a comma-delimited label text file, and a label database file. These labels can be opened in the software of your choice. Router-CIM Automation Suite installation disk includes [RCIM Labeling](#).

This folder will also contain the Touch-N-Print file. The file will end with **.tnp**. This is the file that you would open with the Touch-N-Print label software from CIM-Tech.

The folder will display only if you the selection was made to show this folder located in File/Settings under the '[Systems Folder](#)' tab.

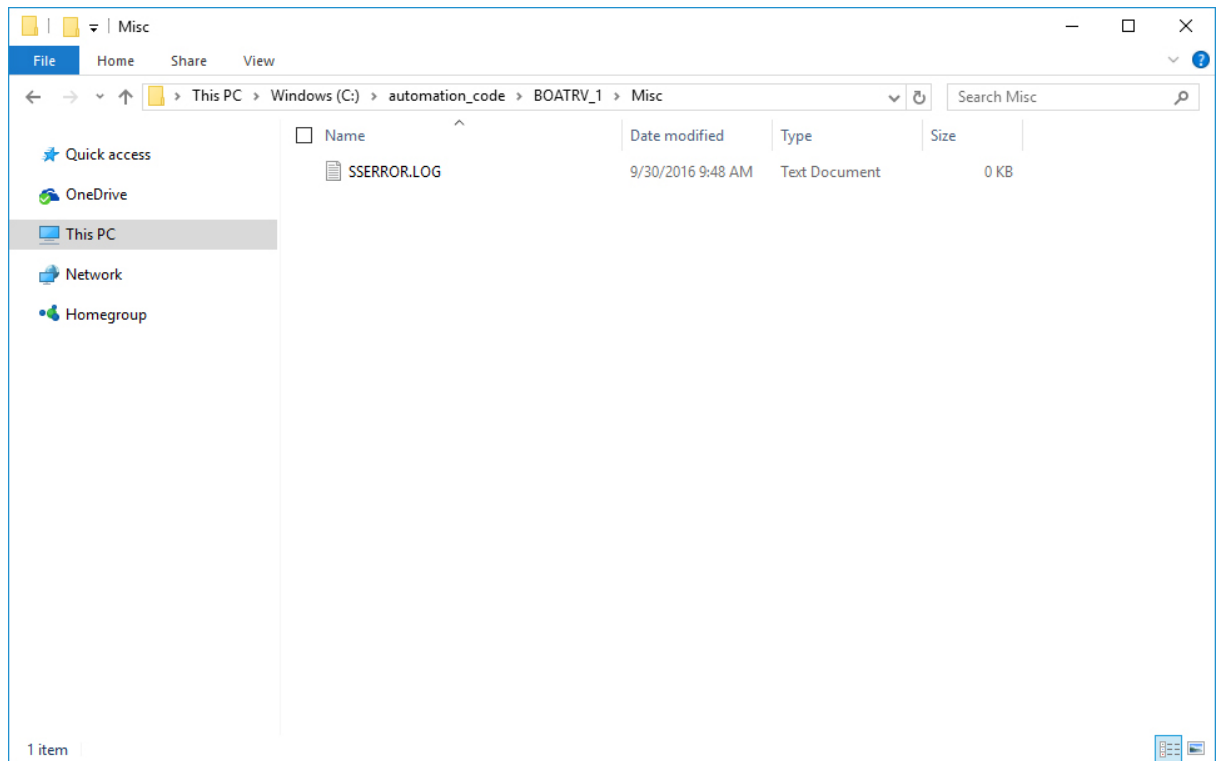


Misc Subfolder

Typically this folder contains the error log and any other miscellaneous files created by the job.

It is usually empty if the job ran successfully.

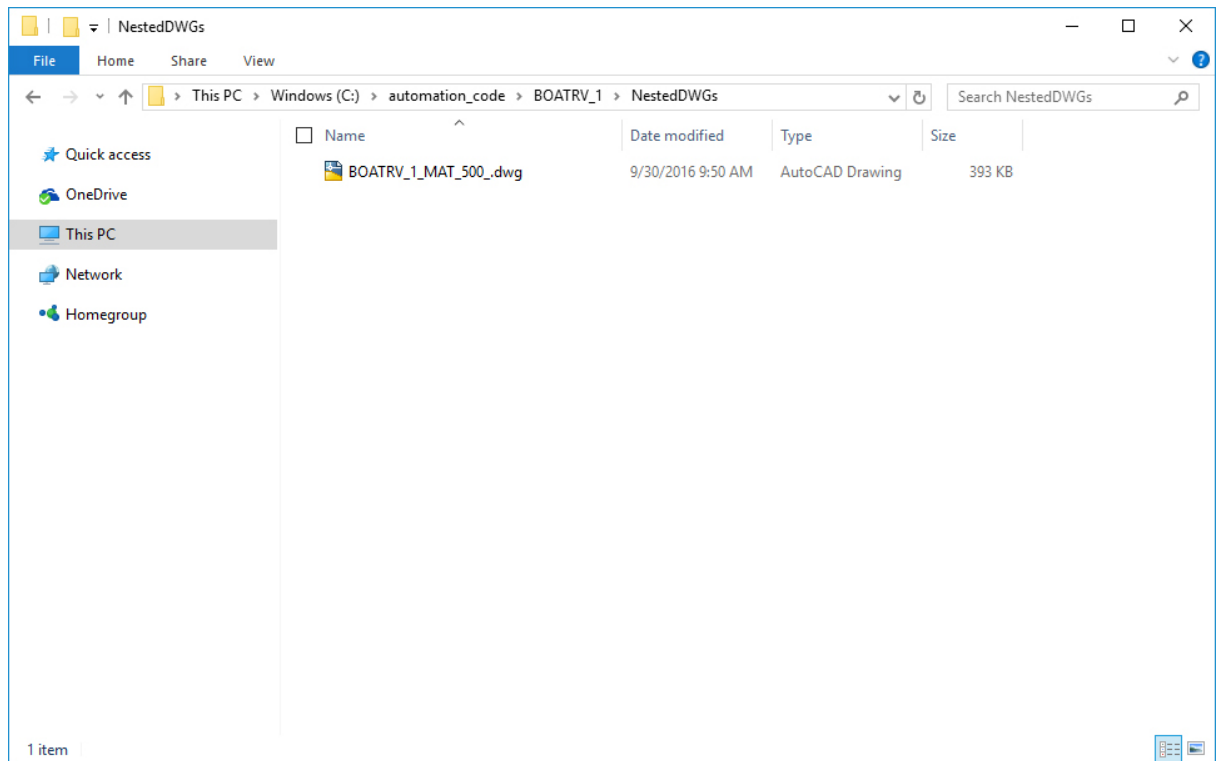
The folder will display only if you the selection was made to show this folder located in File/Settings under the '[Systems Folder](#)' tab.



NestedDWGs Subfolder

This folder contains the nest drawing for all the materials in the job. Each material will have a separate drawing containing all the nests for that material.

The folder will display only if you the selection was made to show this folder located in File/Settings under the '[Systems Folder](#)' tab.



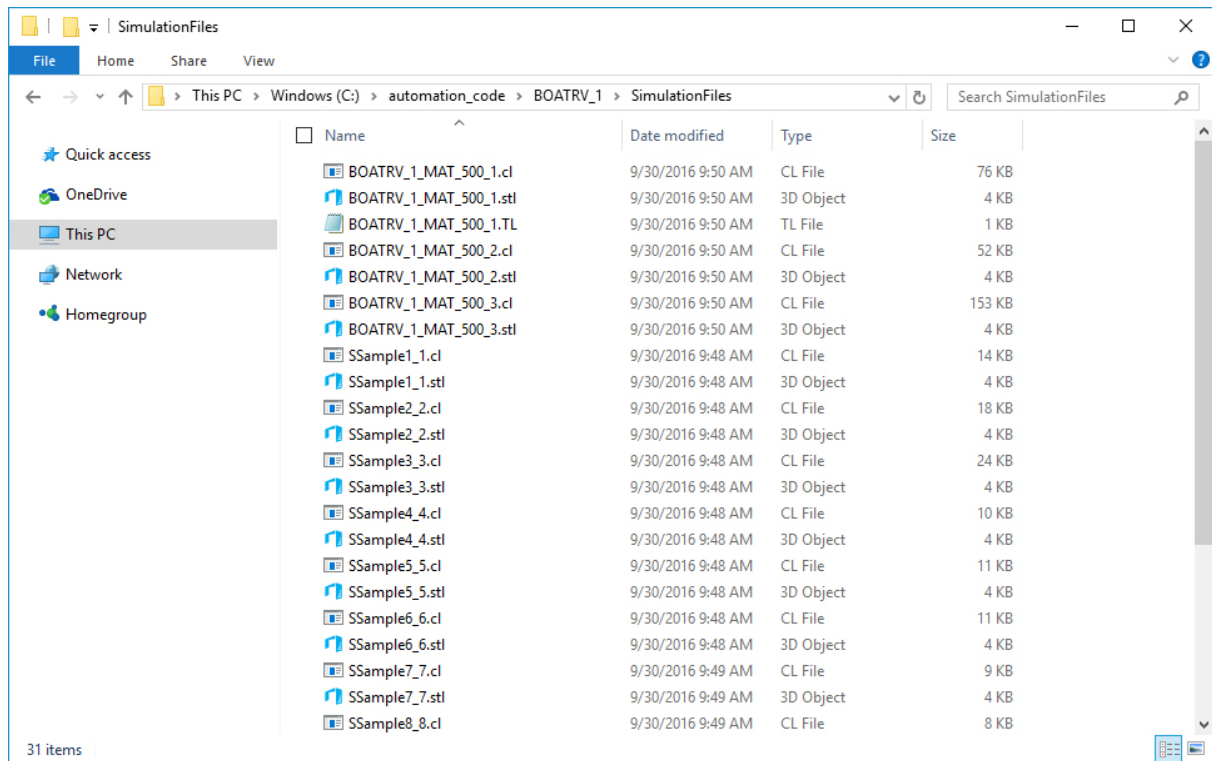
Simulation Files

This folder contains the simulation files needed for CutSIM. Each file will be for a 'Code as Single Part' and for any nests that were created.

The name of the CutSIM files will match your Part Name and Output file settings.

The CutSIM file is the .CL file. The .STL and .TL files are used for the stock and tool definition in the CutSIM program.

Note: This folder will only display if you have the CutSIM software by CIM-Tech.



Summary Report Subfolder

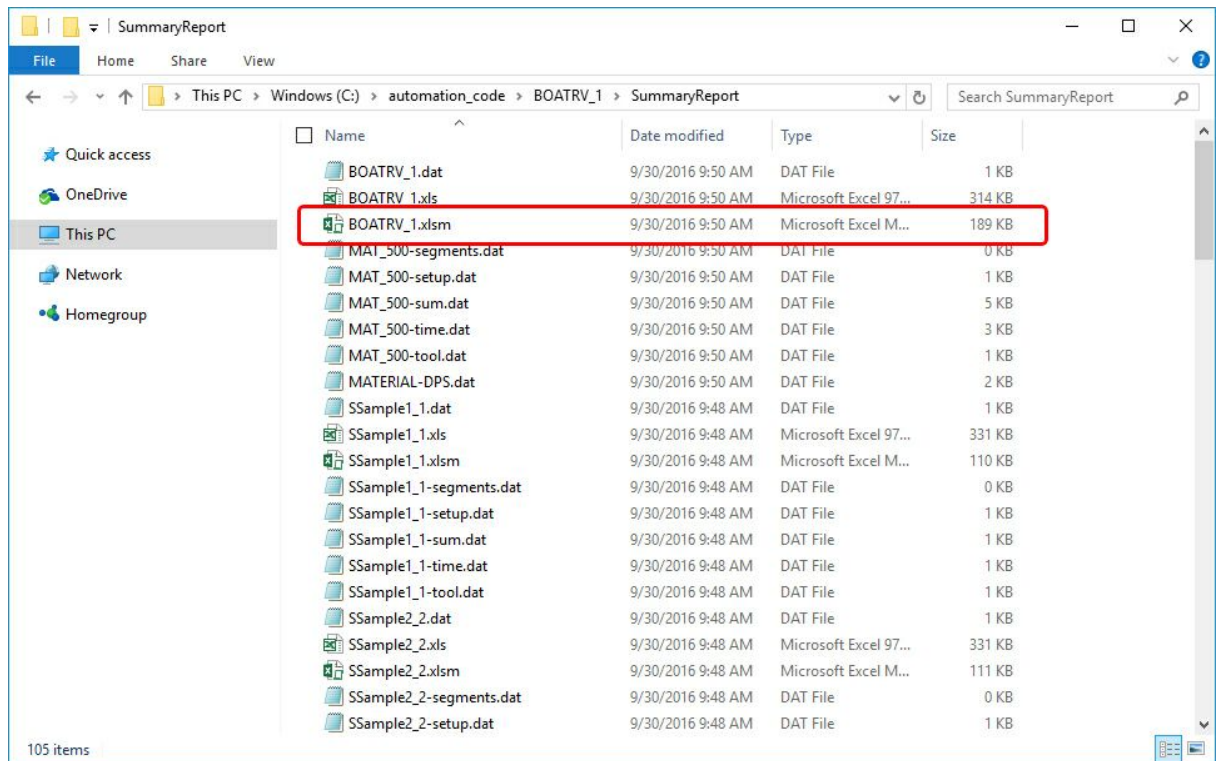
This is optional, but if the **'Create Summary Report'** option is checked in the job under the [Printing and Labeling](#) tab, the summary report data files and Excel spreadsheet will be contained in this folder for both the nested jobs and jobs that were run as 'Code as Single Part'.

You will open the .xlsm file that begins with your Job Name.

The summary report contains information relevant to the job that was run in Router-CIM Automation Suite. Information such as code file names, cycle time estimates, yield, material used, etc.

For more information on the cycle time estimates, [click here](#).

The folder will display only if you the selection was made to show this folder located in File/Settings under the **'Systems Folder'** tab.



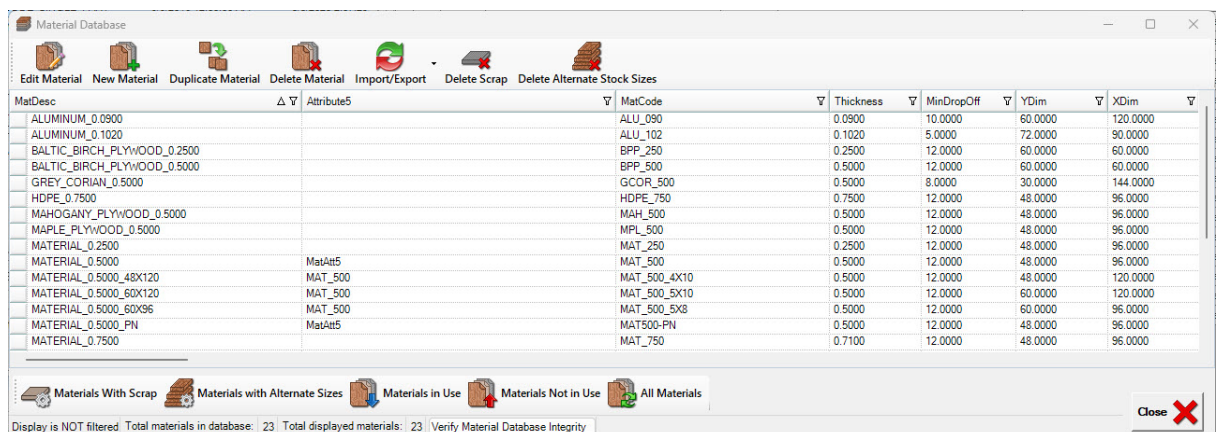
NOTE: Security settings in Excel may need to be adjusted in order to view the Summary Report correctly.

Material Database/Properties

The Materials Database contains a database of all your materials and their individual settings. The Material Database has several options available.

If you double-click on a material in the Materials Database, it will open up the ['Material Properties'](#) window.

At the bottom of the Materials Database, you have options on how to filter the materials that are shown. Select on an option for a further description.



Material Properties (Edit Material)

The Material Properties are what is used to define the material parameters that Router-CIM Automation must follow when the material is used in a job.

There are several default settings that will be present, but the main information that must be entered is:

- Material Description
- Material Code (unique to each material)
- Sheet Stock Y Dim
- Sheet Stock X Dim
- Thickness

Without these, the material cannot be saved.

For more information on ['Scrap Management'](#), select the tab in the image above.

Material Description (Required)

A text field describing the material. This description is most used in the labeling where the material description may be necessary. This field can be up to 255 characters. Any characters are acceptable except quotation marks.

Material Code (Required)

The Material Code is a field that accepts a 25 digit code. This code can contain letters, numbers, underscores (_) and decimal points. This field needs to ALWAYS start with a letter or underscore. You cannot use spaces, and forward or back slashes. Starting the Material Code with a number can result in issues later during job runs under certain circumstances.

Material Handling Code (Optional)

If your machine is equipped with an automatic material load/unload mechanism, you can enter the bunk number where the material is stored here and during the creation of the NC code file, the proper commands will be issued to grab the material from the correct location.

Z0 is top of Material

Typically Z0 is at the top of the spoilboard (bottom of part). Router-CIM Automation Suite automatically shifts all the tool paths and code to allow for the material thickness so that the tool paths are made properly for the material in the job. However, if you touch off your tools to the top of the material instead, check this box to avoid Router-CIM Automation Suite shifting all the tool paths.

Stock Settings

Stock Settings	
Sheet Stock Y Dim	49.0000
Quantity	9999
Sheet Stock X Dim	97.0000
Priority	5
Thickness	0.7500
Bridge Width	0.6250
Cost	0.00

Sheet Stock Y Dim

The dimension of the material in the Y direction on the machine. This is always a positive number.

Sheet Stock X Dim

The dimension of the material in the X direction on the machine. This is always a positive number.

Thickness

The thickness of the material. This is always a positive number.

Bridge Width

This is the minimum distance to keep between parts when nested on the sheet. Typically this is set to at least the largest tool diameter in the job being run.

Note: To adjust the bridge width for a specific knowledge instead of the entire nest, click here to learn about ['Cutter Bridge'](#).

Cost

You may specify the cost of the material in this location. If you have advanced nesting, then you can also add multiple materials of the same type and list their cost, and allow Router-CIM Automation Suite to use the material sized based on cost per job instead of yield only.

Quantity

You may specify a number of sheets on hand in this location. The maximum number of sheets is 999. If you specify a lower number and the job runs out of material, you will be alerted that no suitable stock sizes are available.

Priority

You may specify a priority number from 1-10 for a material, and also for sub-materials of the same type. If you have advanced nesting, you can sort by priority, allowing the use of specific sizes of material as a preference.

Edge Allowance

You can specify areas to leave empty and have the parts offset from by filling in the edge allowances for each side of the sheet. Router-CIM Automation Suite will not nest any parts in this area. This is useful to keep parts for lining up exactly on an unfinished edge of a sheet and allow for any defects that may need to be avoided.



Left Edge Allowance

Default minimum distance to leave empty from the left side of the sheet to the nested parts.

Right Edge Allowance

Default minimum distance to leave empty from the right side of the sheet to the nested parts.

Top Edge Allowance

Default minimum distance to leave empty from the top side of the sheet to the nested parts.

Bottom Edge Allowance

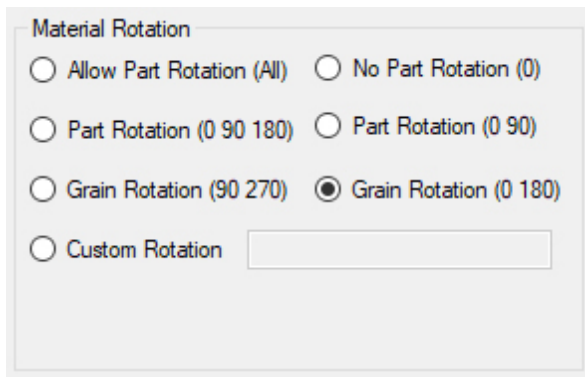
Default minimum distance to leave empty from the bottom side of the sheet to the nested parts.

Irregular Edge Allowance

Default minimum distance to leave empty from the any side of the sheet to the nested parts.

Material Rotation

Material rotation controls how parts are allowed to be placed on the nested sheet.

A screenshot of a 'Material Rotation' dialog box. It contains several radio button options: 'Allow Part Rotation (All)', 'No Part Rotation (0)', 'Part Rotation (0 90 180)', 'Part Rotation (0 90)', 'Grain Rotation (90 270)', 'Grain Rotation (0 180)' (which is selected), and 'Custom Rotation' followed by a text input field.**Allow Part Rotation (All)**

Rotation set to all will allow parts to be rotated at any angle from 0-360° in 1° increments. See [Part Properties](#) under the Part Orientation section for how to override for individual part rotation considerations.

No Part Rotation (0)

Rotation set to 0 will not allow a part to be rotated from the angle it is drawn in. See [Part Properties](#) under the Part Orientation section for how to override for individual part rotation considerations.

Part Rotation (0 90)

Rotation set to 0 or 90 only from the angle it is drawn in. See [Part Properties](#) under the Part Orientation section for how to override for individual part rotation considerations.

Part Rotation (0 90 180)

Rotation set to 0, 90 or 180 only from the angle it is drawn in. See [Part Properties](#) under the Part Orientation section for how to override for individual part rotation considerations.

Grain Rotation (0 180)

Rotation for consideration of grain will only allow a part to be rotated 180° on a nested sheet so that it is aligned with the grain of the material. Consideration should be given to the orientation of the part drawing when considering grain rotation. See [Part Properties](#) under the Part Orientation section for how to override for individual part rotation considerations.

Grain Rotation (90 270)

Rotation for consideration of grain will allow a part to be rotated 90° or 270° on a nested sheet so that it is aligned with the grain of the material for cross-grain. Consideration should be given to the orientation of the part drawing when considering grain rotation. See [Part Properties](#) under the Part Orientation section for how to override for individual part rotation considerations.

Custom Rotation

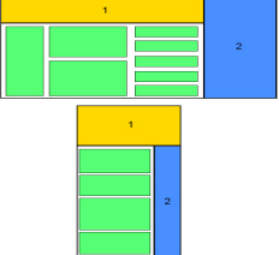
Rotation angle can be defined by the user. See [Part Properties](#) under the Part Orientation section for how to override for individual part rotation considerations.

Material Attributes

You can specify material attributes that you want to have saved with the specific material. The headings of these fields can be customized in the File/Settings under the Field Captions tab.

Material Attributes	
Underscore	<input type="text"/>
Two	<input type="text"/>
Three	<input type="text"/>
Four	<input type="text"/>
Five	<input type="text"/>
Six	<input type="text"/>
Seven	<input type="text"/>
Eight	<input type="text"/>
Nine	<input type="text"/>
Ten	<input type="text"/>

Scrap Management

Scrap Management		Advanced Nesting Parameters											
<input checked="" type="checkbox"/> Make a Scrap Cut <input type="checkbox"/> Inventory Scrap <input type="checkbox"/> Use Scrap <input type="checkbox"/> Inventory to other Material		Scrap Cut Properties <input type="radio"/> Longest Side Only <input type="radio"/> Shortest Side Only <input checked="" type="radio"/> Shortest Side First <input type="radio"/> Longest Side First <input type="radio"/> Vertical Only <input type="radio"/> Horizontal Only <input type="radio"/> Vertical First <input type="radio"/> Horizontal First <input type="radio"/> Combined											
Scrap <div> <input checked="" type="button"/> Add Scrap <input checked="" type="button"/> Delete Scrap </div> <table border="1"> <thead> <tr> <th>YDim</th> <th>XDim</th> <th>Qty</th> <th>Priority</th> <th>Cost</th> </tr> </thead> <tbody> <tr><td> </td><td> </td><td> </td><td> </td><td> </td></tr> </tbody> </table>		YDim	XDim	Qty	Priority	Cost						<div>  </div> <div> Min. X: <input type="text" value="12.0000"/> Min. Y: <input type="text" value="12.0000"/> Distance from Part in X: <input type="text" value="0.3750"/> Distance from Part in Y: <input type="text" value="0.3750"/> Scrap Cut Extension Distance: <input type="text" value="0.3750"/> </div>	
YDim	XDim	Qty	Priority	Cost									

Make a Scrap Cut

Using this setting will instruct Router-CIM Automation Suite to create a scrap tool path.

Note: There must be a knowledge named **SCRAP** in the knowledge drawing. No association is needed in the DOIT file.

If you need a different scrap knowledge for each material, you can make a layer / knowledge association formatted like this:

Knowledge name: **MATERIAL CODE+SCRAP**

Layer name: **MATERIAL CODE+SCRAP**

As an example, The ¾ MDF material has a material code of 75MDF and scrap cuts on this material will use a knowledge called 75MDFSCRAP and the scrap geometry will be on a layer called 75MDFSCRAP.

Inventory Scrap

Setting this option will allow Router-CIM Automation Suite to add newly cut scrap pieces into the materials scrap inventory. You can manually add or delete scrap by selecting the piece of scrap and using the appropriate button.

Use Scrap

Setting this option will force Router-CIM Automation Suite to use scrap material in the job if it is available and of suitable size. Nesting on the scrap piece will follow the material parameters specified.

Inventory to other Material

Setting this option will allow Router-CIM Automation Suite to track the inventory of a specific material but the inventory will be transferred to a different material.

To use this feature, check the box for 'Inventory to other Material'

Scrap Management Advanced Nesting Parameters

☒ Make a Scrap Cut ☐ Inventory Scrap ☐ Use Scrap

☒ Inventory to other Material Inventory to: | ▼

☐ Swap X & Y Axis

Select the material that you would like the scrap pieces to be inventoried to.

Scrap Management Advanced Nesting Parameters

☒ Make a Scrap Cut ☐ Inventory Scrap ☐ Use Scrap

☒ Inventory to other Material Inventory to: | ▼

MatCode	MatDesc	MaterialID	Thickness
ALU_090	ALUMINUM_0.09	414	0.0900
ALU_102	ALUMINUM_0.10	408	0.1020
BPP_500	BALTIC_BIRCH_	403	0.5000
GCOR_500	GREY_CORIAN_	464	0.5000
HDPE_750	HDPE_0.7500	463	0.7500
MAH_500	MAHOGANY_PL	406	0.5000
MPL_500	MAPLE_PLYWO	404	0.5000
MAT_250	MATERIAL_0.250	455	0.2500

Define if you want Router-CIM Automation Suite to Swap X & Y Axis when it is inventoried (rotate the scrap 90 degrees).

Scrap Management Advanced Nesting Parameters

☒ Make a Scrap Cut ☐ Inventory Scrap ☐ Use Scrap

☒ Inventory to other Material Inventory to: | ▼

☐ Swap X & Y Axis

Add Scrap

If you decide not to have Router-CIM Automation Suite inventory scrap or you need to add a piece of scrap material, you can use the 'Add Scrap' button and fill out the appropriate parameters.

Scrap Management Advanced Nesting Parameters

☒ Make a Scrap Cut ☐ Inventory Scrap ☐ Use Scrap
☐ Inventory to other Material

Scrap

 Add Scrap  Delete Scrap

YDim	XDim	Qty	Priority	Cost
49.0000	50.0000	10	1	1.00

Scrap Cut Properties

☐ Longest Side Only
☐ Shortest Side Only
☒ Shortest Side First
☐ Longest Side First
☐ Vertical Only
☐ Horizontal Only
☐ Vertical First
☐ Horizontal First
☐ Combined



Min. X: 12.0000
 Min. Y: 12.0000
 Distance from Part in X: 0.3750
 Distance from Part in Y: 0.3750
 Scrap Cut Extension Distance: 0.3750

Delete Scrap

If you decide to have Router-CIM Automation Suite inventory scrap and you find that a piece of scrap that was inventoried no longer exists, you can use the 'Delete Scrap' button to remove it from the list.

Scrap Management Advanced Nesting Parameters

☒ Make a Scrap Cut ☐ Inventory Scrap ☐ Use Scrap
☐ Inventory to other Material

Scrap

 Add Scrap  Delete Scrap

YDim	XDim	Qty	Priority	Cost
49.0000	50.0000	10	1	1.00

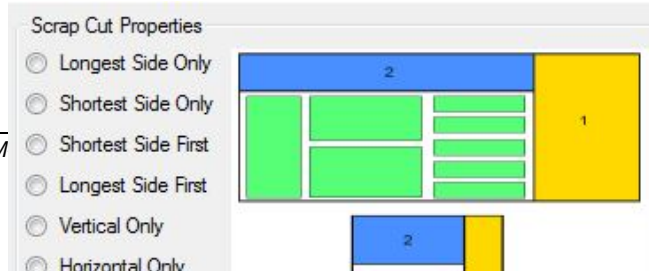
Scrap Cut Properties

☐ Longest Side Only
☐ Shortest Side Only
☒ Shortest Side First
☐ Longest Side First
☐ Vertical Only
☐ Horizontal Only
☐ Vertical First
☐ Horizontal First
☐ Combined



Min. X: 12.0000
 Min. Y: 12.0000
 Distance from Part in X: 0.3750
 Distance from Part in Y: 0.3750
 Scrap Cut Extension Distance: 0.3750

The options for the scrap cut are as follows:

Longest Side Only:**Shortest Side Only:****Shortest Side First:****Longest Side First:****Vertical Only:****Horizontal Only:****Vertical First:****Horizontal First:**

The following parameters need to be filled in depending on the scrap option selected:

The screenshot shows the 'Scrap Management' and 'Advanced Nesting Parameters' tabs. The 'Scrap Management' tab includes checkboxes for 'Make a Scrap Cut', 'Inventory Scrap', and 'Use Scrap', along with a 'Scrap' table with columns for YDim, XDim, Qty, Priority, and Cost. The 'Advanced Nesting Parameters' tab includes 'Scrap Cut Properties' with radio buttons for 'Longest Side Only', 'Shortest Side Only', 'Longest Side First', 'Vertical Only', 'Horizontal Only', 'Vertical First', 'Horizontal First', and 'Combined'. A diagram shows a scrap cut with dimensions 1 and 2. A red box highlights the 'Min. X', 'Min. Y', 'Distance from Part in X', 'Distance from Part in Y', and 'Scrap Cut Extension Distance' fields, which are all set to 0.3750.

Min. X

This parameter is the minimum amount of material left, in units from the right edge of the sheet, required in order to save the scrap and make a scrap cut. If there is less material than this free (from the right edge) then the scrap will not be cut off.

Min. Y

This parameter is the minimum amount of material left, in units from the top edge of the sheet, required in order to save the scrap and make a scrap cut. If there is less material than this free (from the top edge) then the scrap will not be cut off.

Distance from Part in X

This setting is the distance from the edge of the furthest right hand part to the scrap cut off in current units. Typically this is where you can allow for the radius of the largest tool to cut out the parts and further allow for space to cut off the scrap.

Distance from Part in Y

This setting is the distance from the edge of the furthest top part to the scrap cut off in current units. Typically this is where you can allow for the radius of the largest tool to cut out the parts and further allow for space to cut off the scrap.

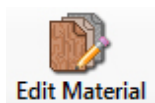
Scrap Cut Extension Distance

This setting is the distance from the edge of the material to the scrap cut off in current units. Typically this is where you can allow for the radius of the largest tool to cut out the parts and further allow for space to cut off the scrap.

Scrap Manager

The Scrap Manager is where you can add, delete, modify scrap pieces in the scrap inventory of the current material. These settings only affect the scrap of the current material.

Edit Material



When you select the **'Edit Material'** button, the currently selected material's properties screen will open allowing you to modify the material's properties.

Material Properties

Material Description: MDF_0.7500

Material Code: MDF_750 Material Handling Code: [] 20 is top of Material

Stock Settings

Sheet Stock Y Dim: 49.0000 Quantity: 9999
 Sheet Stock X Dim: 97.0000 Priority: 5
 Thickness: 0.7500
 Bridge Width: 0.6250
 Cost: 0.00

Edge Allowances

Top: 0.2500
 Left: 0.2500 Right: 0.2500
 Bottom: 0.2500
 Irregular Stock Edge Allowance: 0.2500

Material Rotation

☒ Allow Part Rotation (AR) ☐ No Part Rotation (N)
☐ Part Rotation (0 90 180) ☐ Part Rotation (0 90)
☐ Cross Grain (90 270) ☐ Grain Rotation (0 180)
☐ Custom Rotation

Material Attributes

Underscore: [] Six: []
 Two: [] Seven: []
 Three: [] Eight: []
 Four: [] Nine: []
 Five: [] Ten: []

Scrap Management Advanced Nesting Parameters

☒ Make a Scrap Cut ☐ Inventory Scrap ☐ Use Scrap
☐ Inventory to other Material

Scrap

☒ Add Scrap ☒ Delete Scrap

ID	YDim	XDim	Qty	Δ	Priority	Cost	Bin	Availability

Scrap Cut Properties

☐ Longest Side Only
☐ Shortest Side Only
☒ Shortest Side First
☐ Longest Side First
☐ Vertical Only
☐ Horizontal Only
☐ Vertical First
☐ Horizontal First
☐ Combined

Min. X: 12.0000
 Min. Y: 12.0000
 Distance from Part in X: 0.3750
 Distance from Part in Y: 0.3750
 Scrap Cut Extension Distance: 0.3750

Save Close

Make Default Material For Import Wizards

New



The **'New Material'** option allows you to enter a new material into the database.

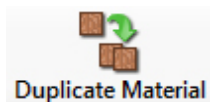
The defaults that a new material has can be defined in the File/Settings under the General Settings tab by defining a **'Default Material for New Materials'**.

Shown above is the default screen for adding a new material to the material database.

There are several default settings that will be present, but the main information that must be entered is the **Material Description**, **Material Code**, **Sheet Y Dim**, **Sheet X Dim** and **Thickness**. Without those, the material cannot be saved.

Each of the settings are described in the [Material Properties](#) section.

Duplicate Material



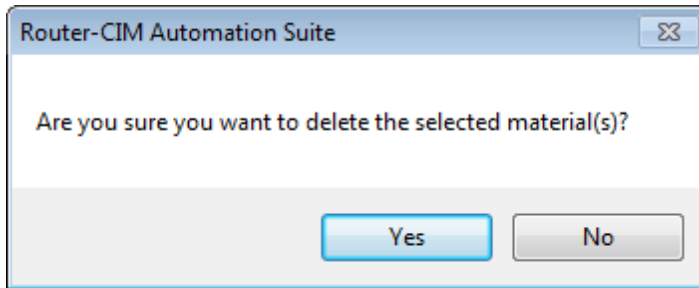
This function will duplicate the currently selected material into a new material so that you can maintain the materials settings. You will need to specify a new Material Description and you **MUST** enter a unique Material Code.

Delete Material

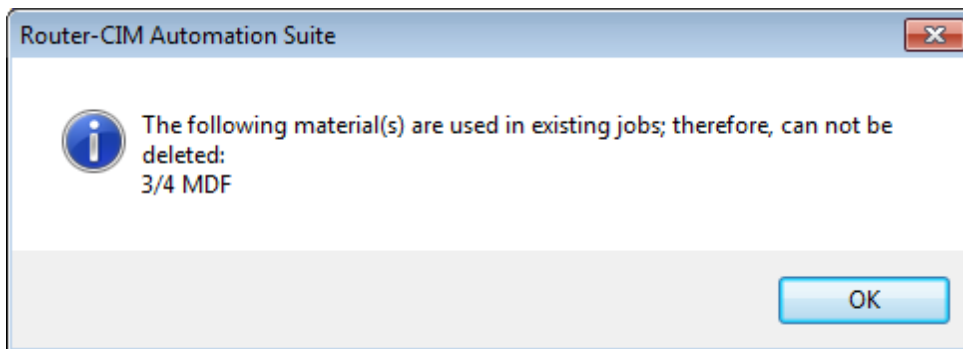


To remove a material from the database, select it in the list and then select the **'Delete Material'** button.

You will be prompted to be sure you want to remove the material.



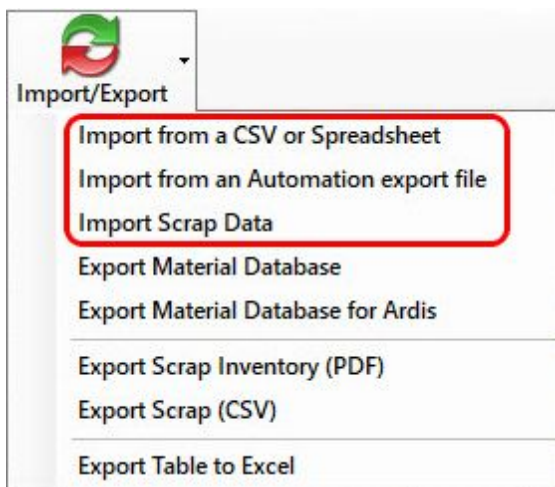
If the material selected is in use in the database on a job in the job tree, you will see the following message:



You then must remove the job that has parts using that material in order to remove the material from the list.

Import Material

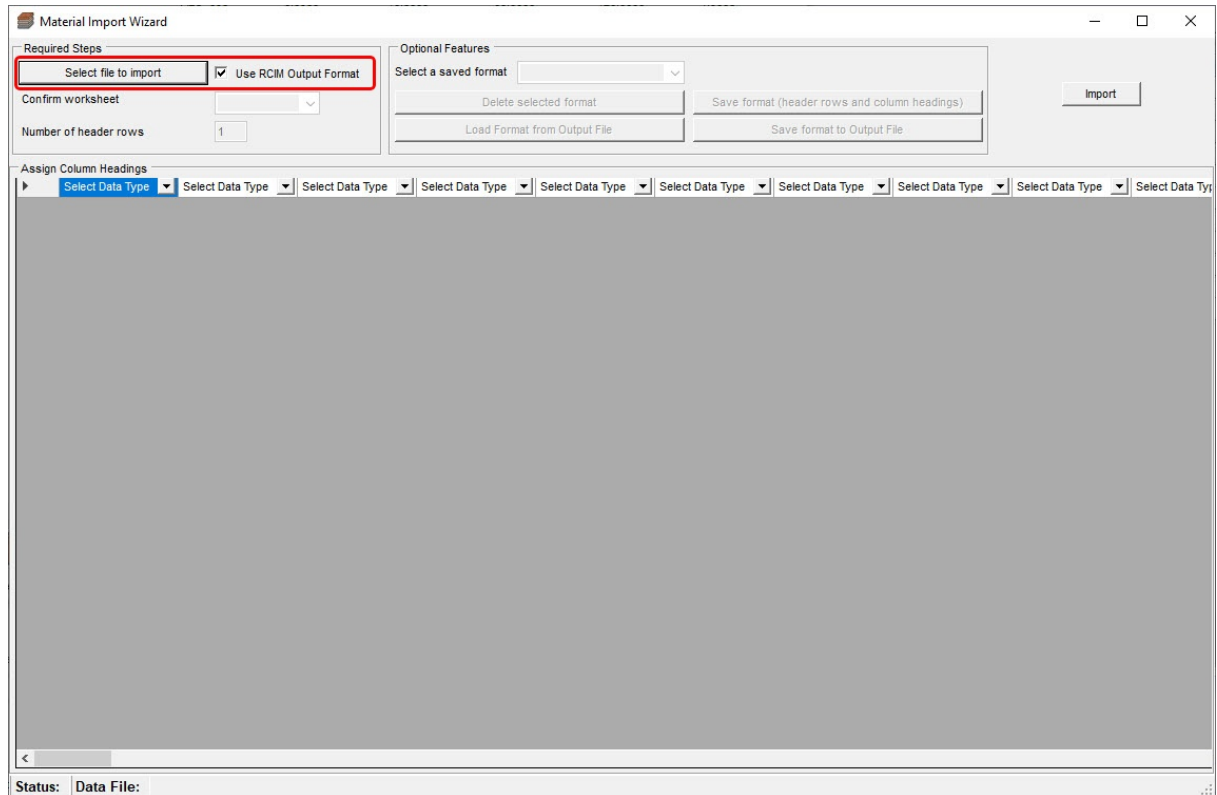
Use this option when bringing materials into Router-CIM Automation Suite from either a Comma Delimited or Spreadsheet file or from an Automation export file.



Import from a CSV or Spreadsheet

This window will allow you to import materials into the material database from a saved Excel Spreadsheet (.xls), Comma Delimited file (.csv). The fields for each column are user definable except in the case of the Automation Material database file, which will fill in the fields automatically.

Selecting the 'Use RCIM Output Format' would preset the columns to match the 'Export Table to Excel' option. If you did not use the 'Export Table to Excel' option, uncheck this feature.

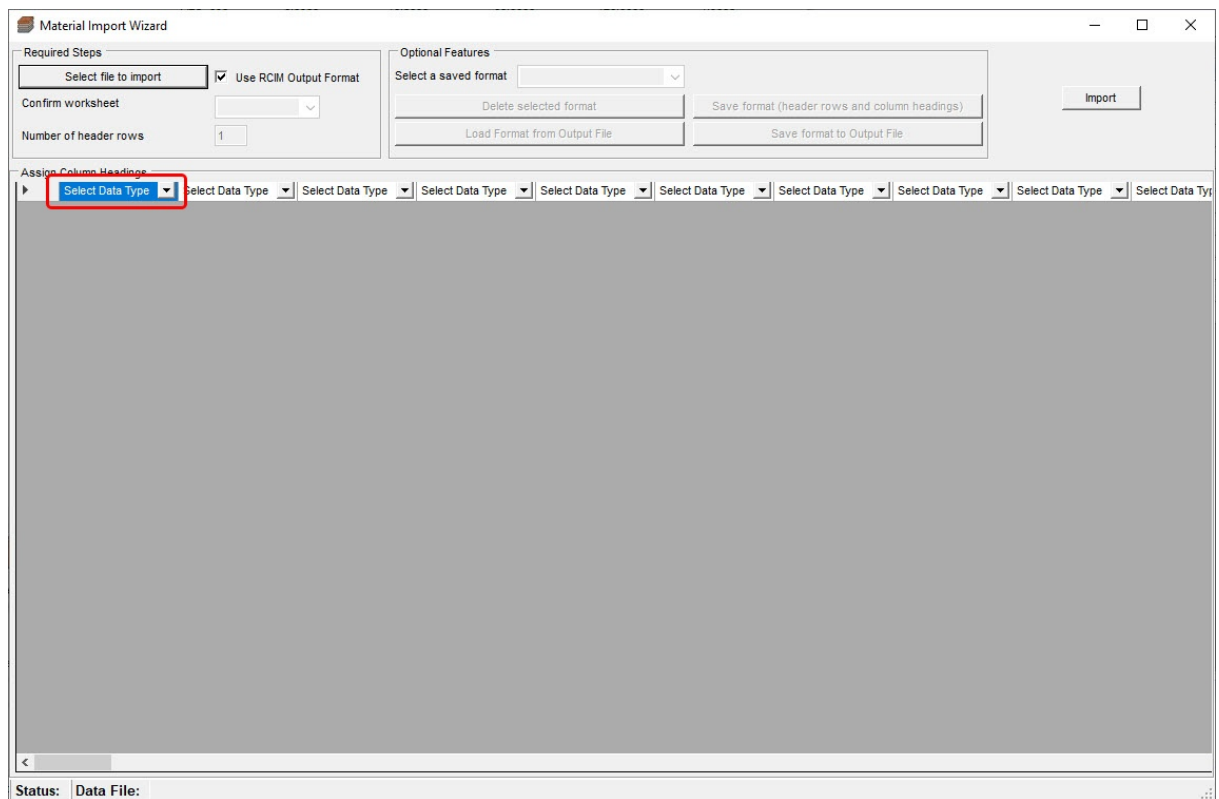


The image shows the 'Material Import Wizard' dialog box. It is divided into several sections:

- Required Steps:** Contains a 'Select file to import' button (highlighted with a red box), a checked checkbox for 'Use RCIM Output Format', a 'Confirm worksheet' dropdown menu, and a 'Number of header rows' input field set to '1'.
- Optional Features:** Contains a 'Select a saved format' dropdown menu, a 'Delete selected format' button, a 'Save format (header rows and column headings)' button, a 'Load Format from Output File' button, and a 'Save format to Output File' button.
- Assign Column Headings:** A large area with a row of dropdown menus, each labeled 'Select Data Type'. The first dropdown is currently set to 'Select Data Type'.
- Status:** A section at the bottom left labeled 'Status: Data File:'.

An 'Import' button is located on the right side of the dialog box.

Once you have selected the correct Excel Spreadsheet (.xls), Comma Delimited file (.csv) from the File Explorer and selected open, you will need to define each column so that Router-CIM Automation Suite will know where to put the information:



The Material Import Wizard dialog box is shown. It has a title bar with a minus, maximize, and close button. The dialog is divided into several sections:

- Required Steps:** Includes a "Select file to import" button, a checked "Use RCIM Output Format" checkbox, a "Confirm worksheet" dropdown, and a "Number of header rows" input field set to 1.
- Optional Features:** Includes a "Select a saved format" dropdown, a "Delete selected format" button, a "Save format (header rows and column headings)" button, a "Load Format from Output File" button, and a "Save format to Output File" button.
- Assign Column Headings:** A section with a row of dropdown menus. The first dropdown is highlighted with a red box and contains the text "Select Data Type". The other dropdowns also contain "Select Data Type".
- Status:** A section at the bottom with a "Data File:" label.

The available data types for each field are:

- Material Description
- Material Code
- X-Dim
- Y-Dim
- Thickness
- Min. Drop Off X
- Min. Drop Off Y
- Bridge Width
- Left Edge Allowance
- Right Edge Allowance
- Top Edge Allowance
- Bottom Edge Allowance
- Irregular Stock Edge Allowance
- Rotation
- Cut Scrap
- Use Scrap
- Inventory Scrap
- Scrap Cut Distance from Part X
- Scrap Cut Distance from Part Y
- Scrap Cut Extension
- Material Handling Code
- Attribute 1
- Attribute 2
- Attribute 3
- Attribute 4

- Attribute 5
- Attribute 6
- Attribute 7
- Attribute 8
- Attribute 9
- Attribute 10
- Z0 is top
- Cost
- Nest Type (0; Priority;1:Price;2:Yield)
- Jobs per Cart
- Open Cart Threshold
- Quantity
- Priority
- Alt. Materials Decrease Quantity
- Ignore
- Disable Scrap
- Scrap Cut Definition
- Skeleton Offset Distance
- Skeleton X Spacing
- Skeleton Y Spacing
- Skeleton Equal Spacing
- Skeleton Equal Spacing X
- Skeleton Equal Spacing Y
- Enable Skeleton
- Inventory to Different Material
- Inventory to Different Material Swap Axis
- Inventory Scrap to Different Material
- Material GUID
- Veneer Match

When you have defined the columns, you can save the column setup by selecting 'Save Format (header rows and column headings)'. This will prompt you to assign a name for the format so that it is available for future imports if needed.

Saved Format Name

Please enter a name for this format.

OK

Cancel

RCIM-Material-Import

The next time you import, you can select the format from the drop-down area.

Material Import Wizard

Required Steps

Select file to import ☐ Use RCIM Output Format ☒

Confirm worksheet

Number of header rows

Optional Features

Select a saved format **RCIM-Material-Import**

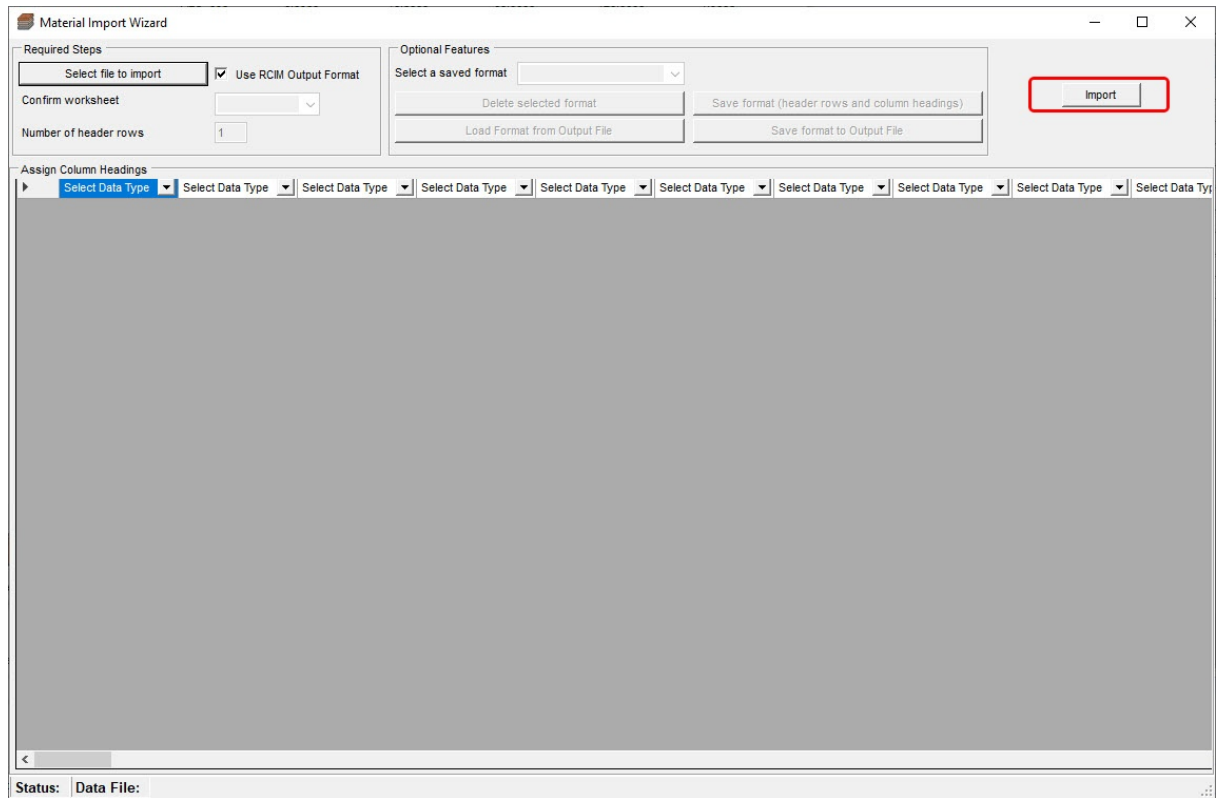
Delete selected format

Save format (header rows and column headings)

Load Format from Output File

Save format to Output File

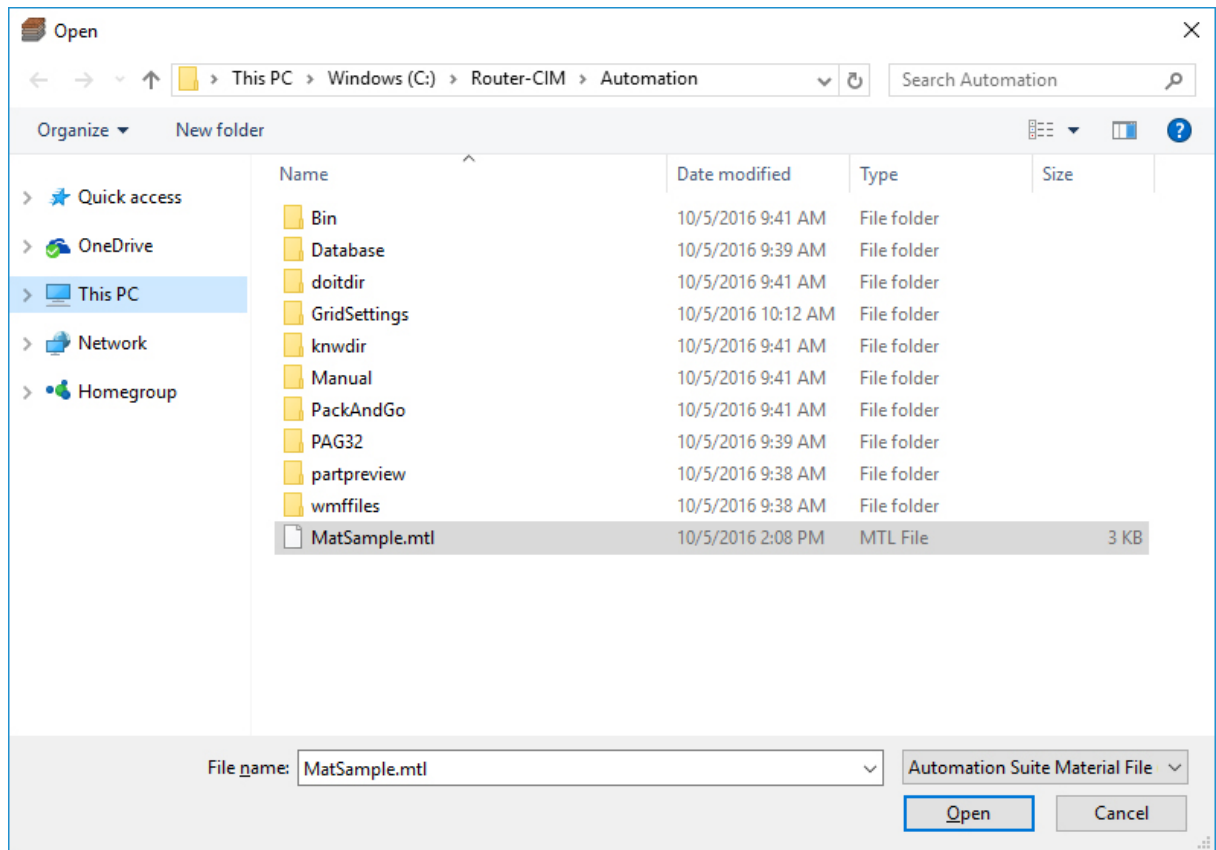
Once the columns have been defined, select the 'Import' button:



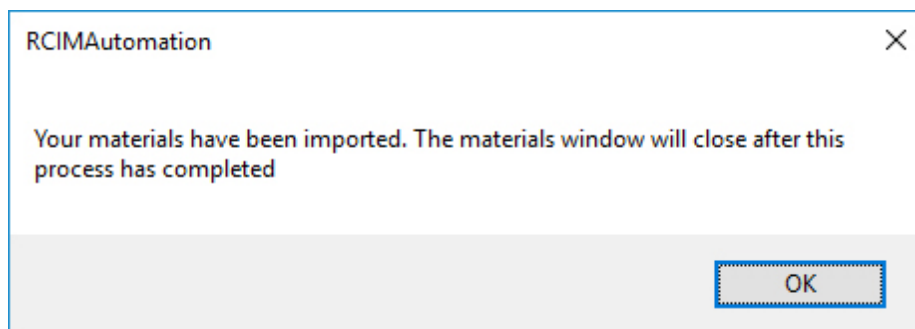
The image shows a 'Material Import Wizard' dialog box. It is divided into several sections. The 'Required Steps' section on the left includes a 'Select file to import' button, a checked 'Use RCIM Output Format' checkbox, a 'Confirm worksheet' dropdown menu, and a 'Number of header rows' input field set to '1'. The 'Optional Features' section on the right includes a 'Select a saved format' dropdown menu, a 'Delete selected format' button, a 'Save format (header rows and column headings)' button, a 'Load Format from Output File' button, and a 'Save format to Output File' button. A red rectangle highlights the 'Import' button in the top right corner. Below these sections is a large area for 'Assign Column Headings', which contains a row of dropdown menus, each labeled 'Select Data Type'. The first dropdown menu is highlighted in blue. At the bottom of the dialog, there is a 'Status: Data File:' label.

Import from an Automation Export File

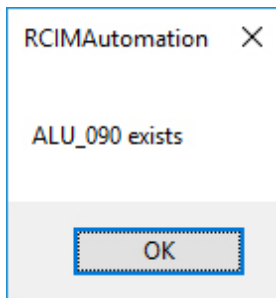
If you are importing from an Automation export file, once selected, a windows browser will open allowing you to select the appropriate .MTL file to import.



Once you select 'Open' Router-CIM Automation Suite will import the MTL file. If no materials need to be overwritten, you will see this:



If the material is already in the database, Router-CIM Automation will overwrite the material currently in the database:



Import Scrap Data

This window will allow you to import scrap materials into the material database from a saved Excel Spreadsheet (.xls), Comma Delimited file (.csv). The fields for each column are user definable.

Choose 'Select File to Import' and select Excel Spreadsheet (.xls), Comma Delimited file (.csv) that you want to import.

The "Scrap Import Wizard" window is shown. It has a "Required Steps" section with "Select file to import" highlighted by a red box. Below it are "Confirm worksheet" and "Number of header rows" (set to 1). To the right, there's a section "Use the following material if no material identifying column is present" with a dropdown menu and a checkbox "Use the above selected material if the material identifying data exists but does not match an existing material in the material database". An "Import Scrap Data" button is on the right. Below these is a table for "Assign Column Headings" with columns: Ignore, MaterialID, YDim, XDim, Qty, Priority, Description, MaterialCode, StockType, Cost, Bin, and Ignore. The first row of data shows: 77, 461, 49, 60, 10, 1, Material_0.7500, MAT_750, 0, 0, 0, and Ignore. The status bar at the bottom shows "Status: Idle" and "Data File: C:\Users\prom4\Desktop\RCIM_Scrap.csv".

Ignore	MaterialID	YDim	XDim	Qty	Priority	Description	MaterialCode	StockType	Cost	Bin	Ignore
77	461	49	60	10	1	Material_0.7500	MAT_750	0	0	0	Ignore

Adjust the column headings and header rows to configure the import wizard.

Scrap Import Wizard

Required Steps

Select file to import

Confirm worksheet

Number of header rows

Use the following material if no material identifying column is present

Use the above selected material if the material identifying data exists but does not match an existing material in the material database

Import Scrap Data

Assign Column Headings

	Ignore	MaterialID	YDim	XDim	Qty	Priority	Description	MaterialCode	StockType	Cost	Bin	Ignore
		461	49	60	10	1	Material_0.7500	MAT_750	0	0	0	

Status: Idle | Data File: C:\Users\prom4\Desktop\RCIM_Scrap.csv

If no Material ID is present, select the material from the drop-down for where Router-CIM should import the scrap material.

Scrap Import Wizard

Required Steps

Select file to import

Confirm worksheet

Number of header rows

Use the following material if no material identifying column is present

☐ Use the above selected material if the material identifying data exists but does not match an existing material in the material database

Import Scrap Data

Assign Column Headings

	Ignore	MaterialID	YDim	XDim	Qty	Priority	Description	MaterialCode	StockType	Cost	Bin	Ignore
77		461	49	60	10	1	Material_0.7500	MAT_750	0	0	0	

Status: Idle | Data File: C:\Users\prom4\Desktop\RCIM_Scrap.csv

Select 'Import Scrap Data' when finished.

Scrap Import Wizard

Required Steps

Select file to import

Confirm worksheet

Number of header rows

Use the following material if no material identifying column is present

☐ Use the above selected material if the material identifying data exists but does not match an existing material in the material database

Import Scrap Data

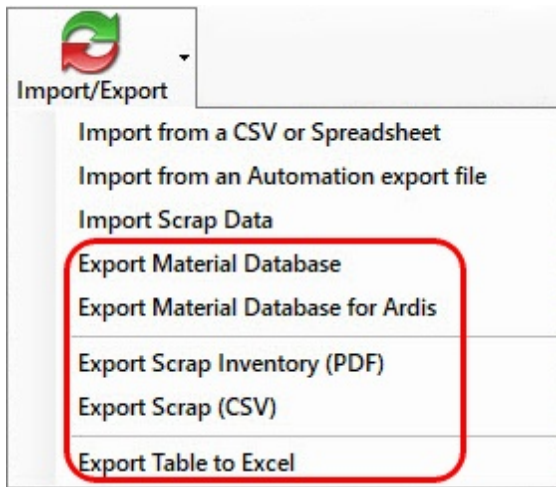
Assign Column Headings

	Ignore	MaterialID	YDim	XDim	Qty	Priority	Description	MaterialCode	StockType	Cost	Bin	Ignore
77		461	49	60	10	1	Material_0.7500	MAT_750	0	0	0	

Status: Idle | Data File: C:\Users\prom4\Desktop\RCIM_Scrap.csv

Export Material

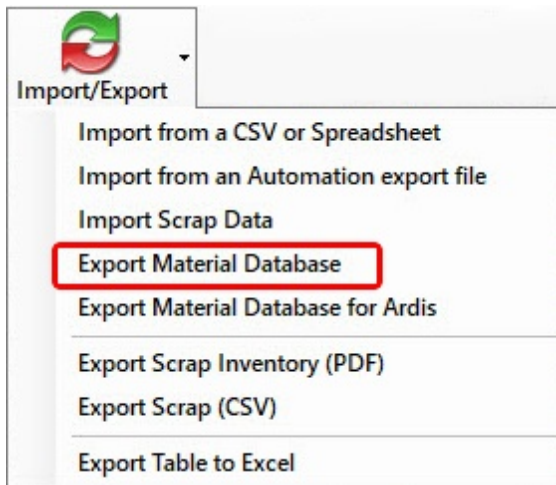
Router-CIM Automation Suite will allow you to export one, several, or all materials to an importable database creating an .MTL file. This allows you to share or back up your material database so that you can import it into another database at a later time, or on another computer.



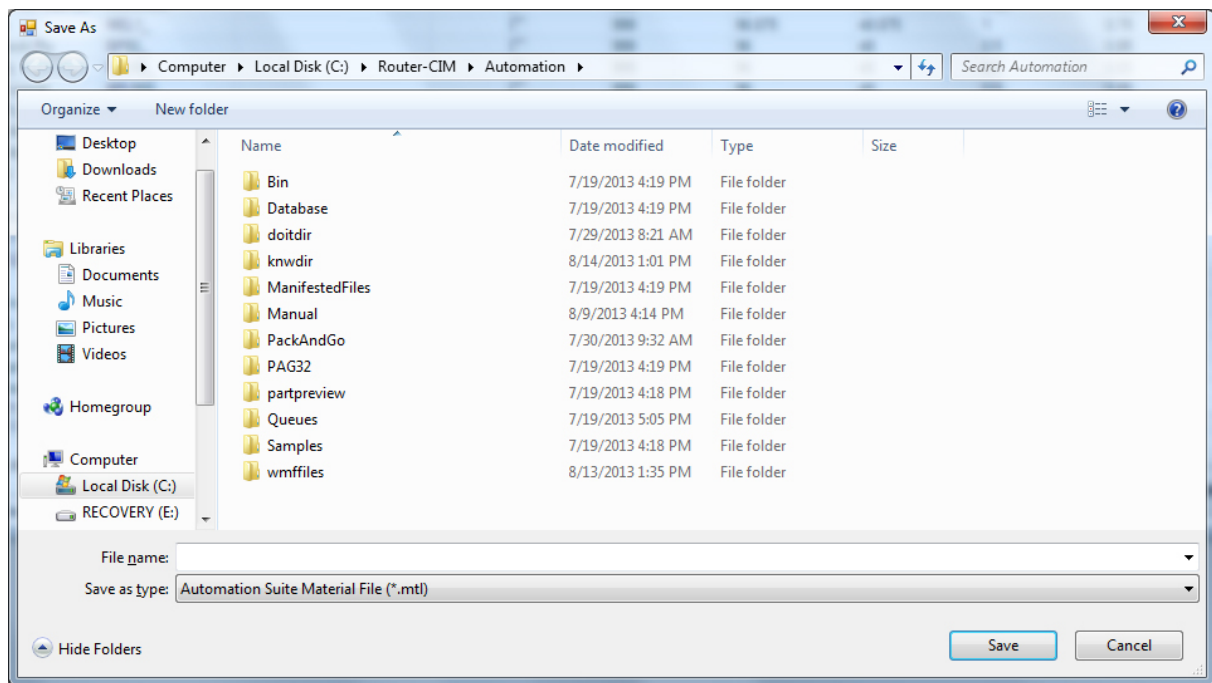
Export Material Database

To export materials, select one or more materials from the list or optionally select All Materials.

Once the materials are selected, select the **'Export Material Database'** option.



You will next be shown a screen where you can select a name and location for the exported materials file to be stored.

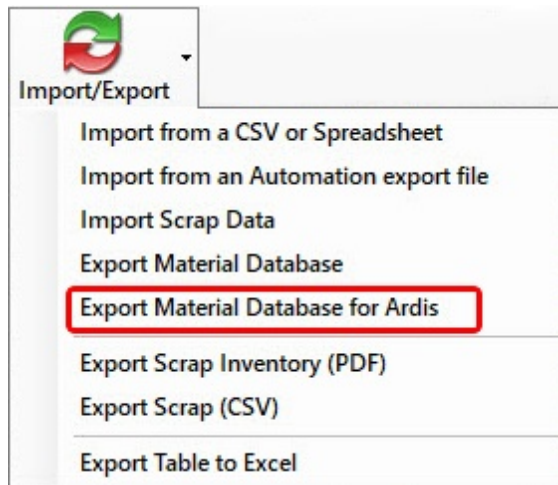


Once you select **'Save'**, the material file will be exported and ready for later import.

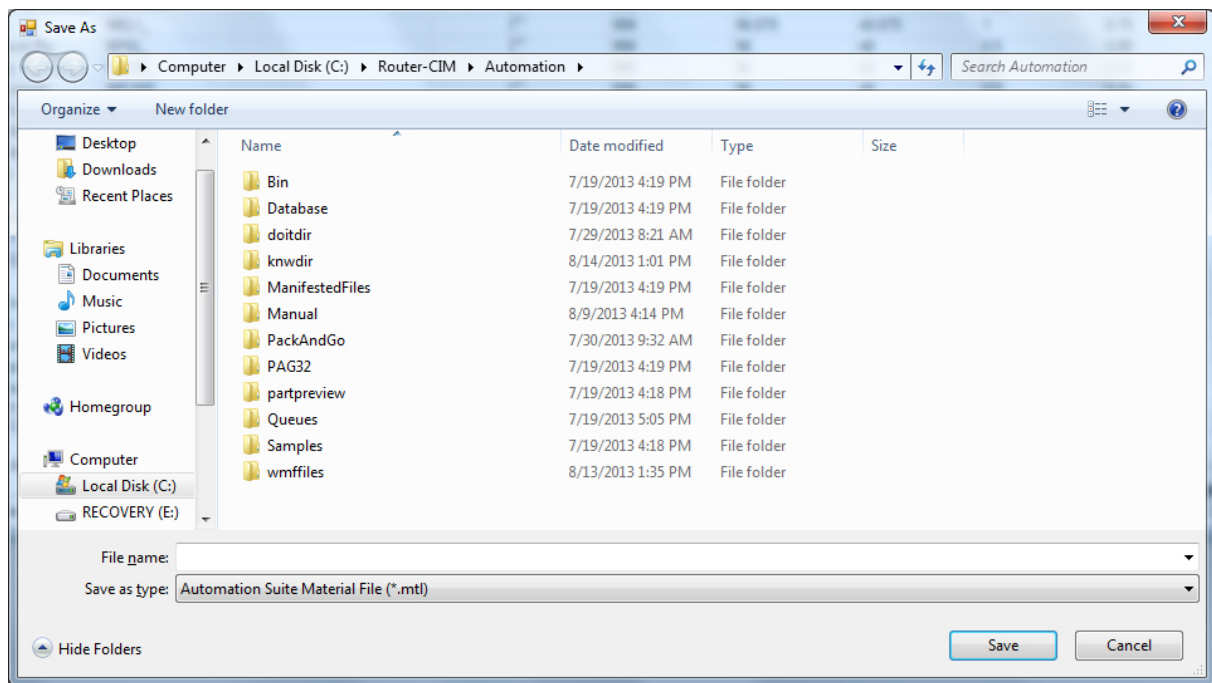
Export Material Database for Ardis

To export materials, select one or more materials from the list or optionally select All Materials.

Once the materials are selected, select the **'Export Material Database for Ardis'** option.



You will next be shown a screen where you can select a name and location for the exported materials file to be stored.



Once you select **'Save'**, the material file will be exported and ready for later import into the Ardis saw optimization software (optional).

Export Scrap Inventory (PDF)

Router-CIM Automation Suite can also export the scrap inventory to a Comma-delimited (CSV) file.



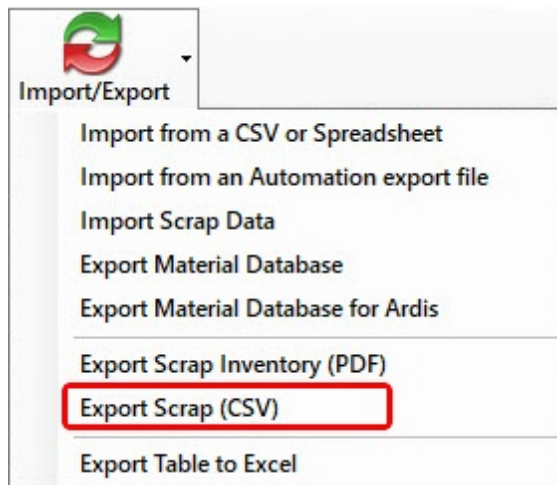
Router-CIM Automation Suite will create a PDF file that will list the scrap entries so they can be checked against what is on hand.

Router-CIM Automation Scrap Inventory Report

Material Desc	Material Code	XDim	YDim	Cost	Quantity
1/2 Baltic Birch Plywood	BP50_	100.0000	66.0000	2.00	1
Material Desc	Material Code	XDim	YDim	Cost	Quantity
3/4MDF5x8	MDF3458	44.0000	35.0000	0.00	2
3/4MDF5x8	MDF3458	40.0000	40.0000	4.0000	4

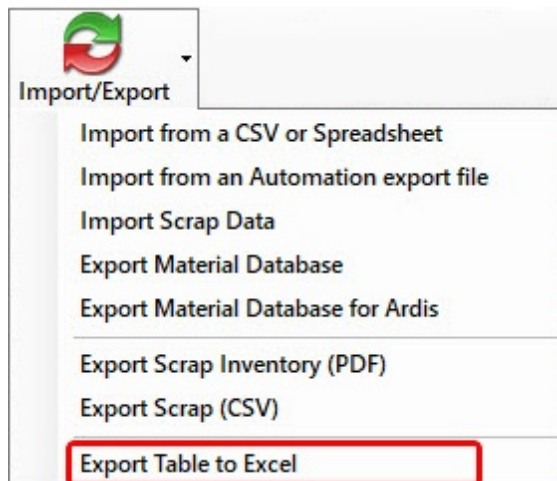
Export Scrap (CSV)

Router-CIM Automation Suite can also export the scrap inventory to a Comma-delimited (CSV) file.



Export Table to Excel

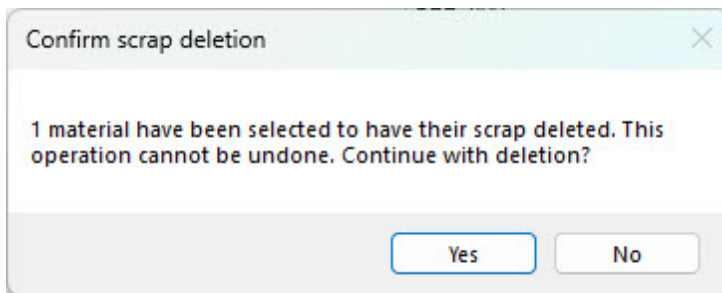
Router-CIM Automation Suite can export the materials to an Excel (.xls) file.



Delete Scrap



You may delete the scrap entries for the selected materials by selecting this option. You will be prompted with the following screen to be sure you wish to delete all the scrap entries for the selected materials. Once the scrap entries are deleted, this operation cannot be undone.



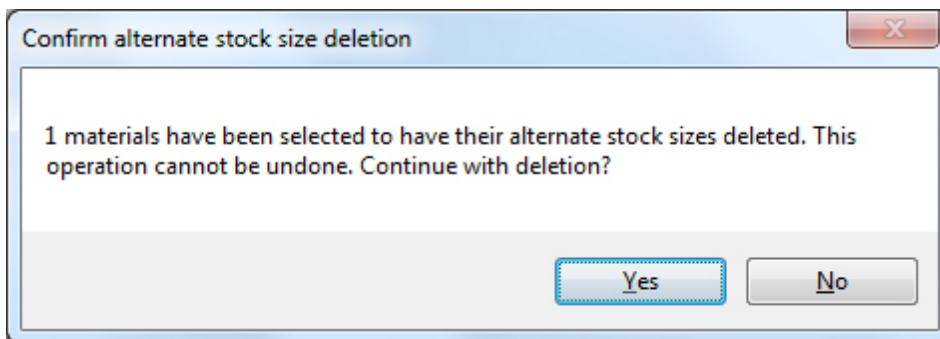
Delete Alternate Sizes



You may delete the alternate sizes for all materials by selecting this option.

Select the materials that you want to delete the alternate sizes for.

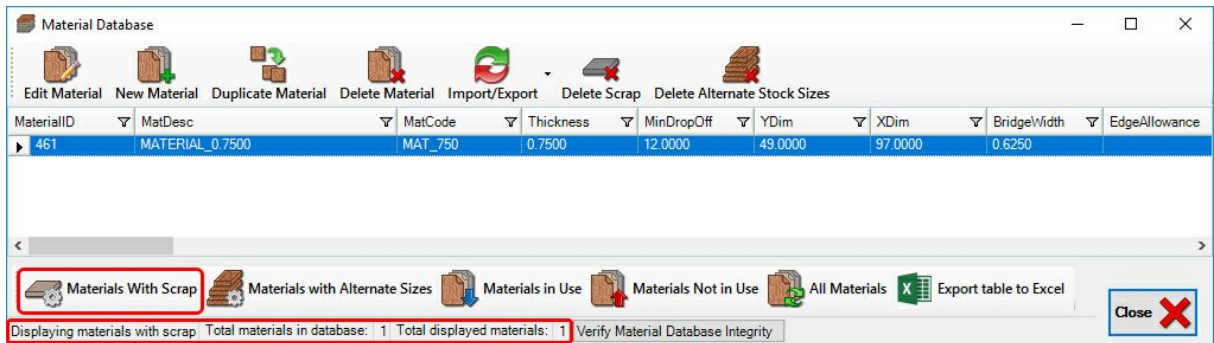
You will be prompted with the following screen to be sure you wish to delete all the scrap entries in the database.



Once the alternate size entries are deleted, you cannot get them back!

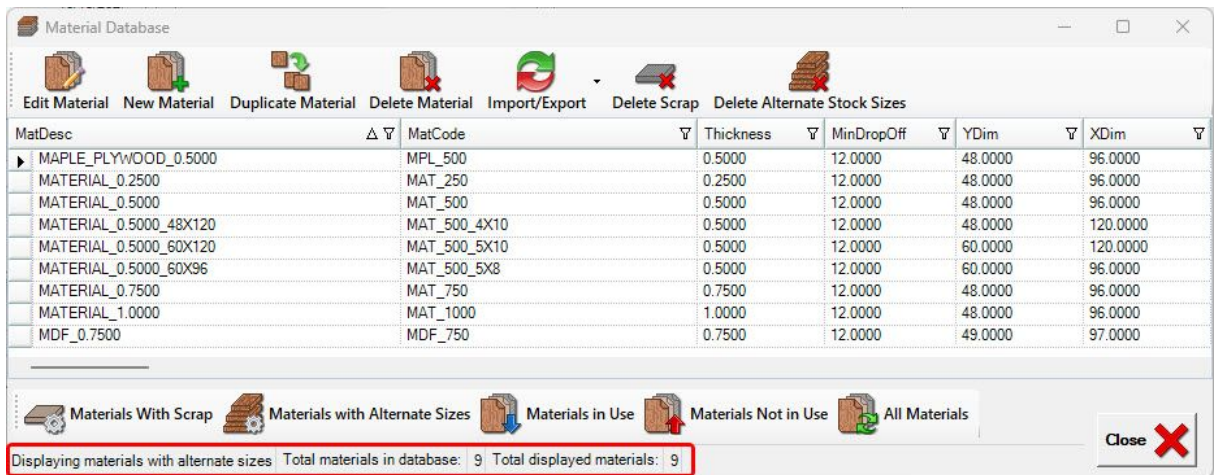
Materials with Scrap

Selecting the **'Materials with Scrap'** button will show all materials in the database that currently have scrap material associated with them.



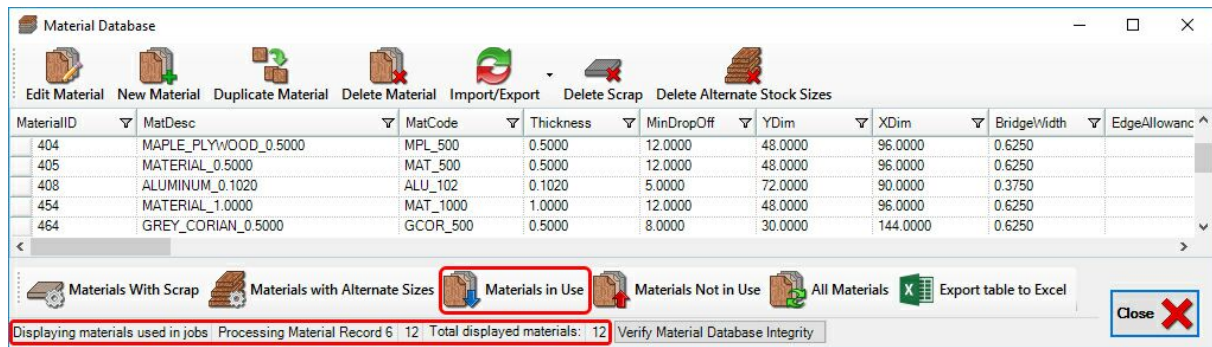
Materials with Alternate Sizes

Selecting the **'Materials with Alternate Sizes'** button will show all materials in the database that currently have alternate sizes associated with them.



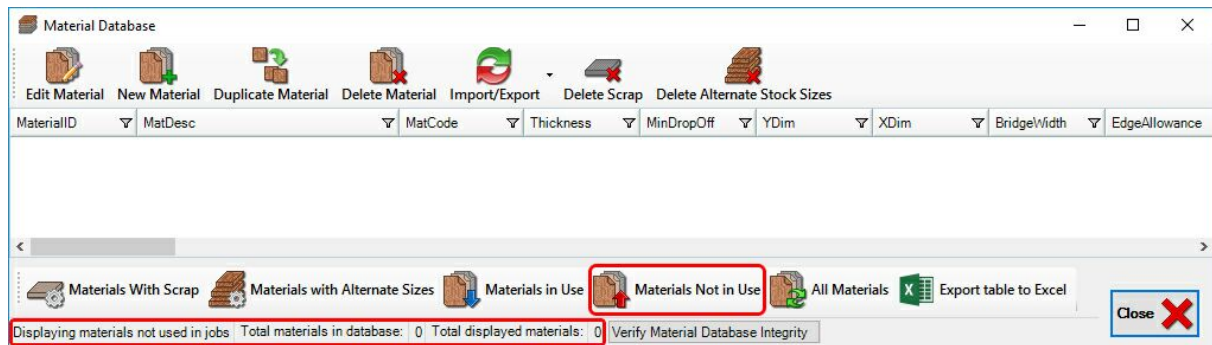
Materials in Use

Selecting the **'Materials in Use'** button will show all materials in the database that are currently have a parts in a job associated with them.



Materials Not in Use

Selecting the '**Materials Not in Use**' button will show all materials in the database that are currently have NO parts in a job associated with them.



All Materials

Selecting the '**All Materials**' button will show all materials in the database.

MaterialID	MatDesc	MatCode	Thickness	MinDropOff	YDim	XDim	BridgeWidth	EdgeAllowance
414	ALUMINUM_0.0900	ALU_090	0.0900	10.0000	60.0000	120.0000	1.5000	
408	ALUMINUM_0.1020	ALU_102	0.1020	5.0000	72.0000	90.0000	0.3750	
403	BALTIC_BIRCH_PLYWOOD_0.5000	BPP_500	0.5000	12.0000	60.0000	60.0000	0.6250	
464	GREY_CORIAN_0.5000	GCOR_500	0.5000	8.0000	30.0000	144.0000	0.6250	
463	HDPE_0.7500	HDPE_750	0.7500	12.0000	48.0000	96.0000	0.7500	
406	MAHOGANY_PLYWOOD_0.5000	MAH_500	0.5000	12.0000	48.0000	96.0000	0.6250	
404	MAPLE_PLYWOOD_0.5000	MPL_500	0.5000	12.0000	48.0000	96.0000	0.6250	
455	MATERIAL_0.2500	MAT_250	0.2500	12.0000	48.0000	96.0000	0.6250	
405	MATERIAL_0.5000	MAT_500	0.5000	12.0000	48.0000	96.0000	0.6250	
461	MATERIAL_0.7500	MAT_750	0.7500	12.0000	49.0000	97.0000	0.6250	
454	MATERIAL_1.0000	MAT_1000	1.0000	12.0000	48.0000	96.0000	0.6250	
462	MDF_0.7500	MDF_750	0.7500	12.0000	49.0000	97.0000	0.6250	

Materials With Scrap Materials with Alternate Sizes Materials in Use Materials Not in Use All Materials Export table to Excel

Display is NOT filtered Processing Material Record 12 12 Total displayed materials: 12 Verify Material Database Integrity

Advanced Nesting Module - Materials

The Advance Nesting Module (optional) from Router-CIM Automation Suite includes additional features for handling the material database.

Skeleton-Cut-Off Parameters

Enable Skeleton-Cut-Off ☐

Offset Distance

X Spacing

Y Spacing

Equal Spacing ☐

of X Spacings

of Y Spacings

Multi-Stock Nesting

☒ Priority ☐ Best Yield ☐ Best Price

☐ Automatically Decrease Qty

Alternate Sizes

+ Add Alternate Size - Delete Alternate Size

YDim	XDim	Qty	Priority	Cost
49.0000	121.0000	999	8	0.0000
61.0000	97.0000	999	10	0.0000
61.0000	121.0000	999	10	0.0000

☒ Allow Veneer Matching

Cart Control Parameters

Jobs per Cart

Open Cart Threshold

The features are Skeleton Scrap Cut, Multi-Stock Nesting (alternate sheet sizes), Cart Control Parameters and Veneer Matching Parameters.

Skeleton Scrap Cut

The Skeleton Scrap Cutting feature will allow the user to define a maximum size of scrap that they will want the Router-CIM Automation Suite to leave. The benefits of this type of scrap cutting are that the operator will have smaller pieces of scrap to remove from the table. This feature is integrated with Router-CIM Automation Suite's Scrap Management System that allows for multiple direction scrap cuts. You will still be able to take advantage of the inventoried scrap while easing the strain of removing large sections of waste material by cutting them down to a manageable size.

To set up the Skeleton Scrap Cut you will need to define the following parameters:

Note: There must be a knowledge named **SKELETON** in the knowledge drawing. It will need to be a Centerline Ramp Cycle with your defined tooling parameters.

If you need a different knowledge for each material in your job, you can make a layer / knowledge association formatted like this:

Knowledge name: **MATERIAL CODE+SKELETON**

Layer name: **MATERIAL CODE+SKELETON**

As an example, The ¾ MDF material has a material code of 75MDF and the skeleton cuts on this material will use a knowledge called 75MDFSKELETON and the skeleton geometry will be on a layer called 75MDFSKELETON.

Enable Skeleton-Cut-Off

To allow Router-CIM Automation Suite to make a Skeleton Scrap Cut, you will need to check the box. A check in the box indicates that a skeleton cut will be made. No check in the box indicates a skeleton cut will not be made.

Offset Distance:

This parameter defines how far from the edge of the part will Router-CIM Automation Suite create the geometry that the SKELETON knowledge will follow. This distance needs to be at least the radius of the tool.

X Spacing:

Specifies the distance between skeleton cuts along the X axis.

Y Spacing:

Specifies the distance between skeleton cuts along the Y axis.

Equal Spacing:

To allow Router-CIM Automation Suite to make a Skeleton Scrap Cut based on a number of cuts to be made in the X and Y axes, you will need to check the box. A check in the box indicates that a skeleton cut will be made with equal spacing. No check in the box indicates a skeleton cut will be made based on the X and Y spacing defined above.

of X Spacings:

Specifies the number of cuts that will be perpendicular to the X axis.

of Y Spacings:

Specifies the number of cuts that will be perpendicular to the Y axis.

Skeleton-Cut-Off Parameters

Enable Skeleton-Cut-Off ☒

Offset Distance

X Spacing

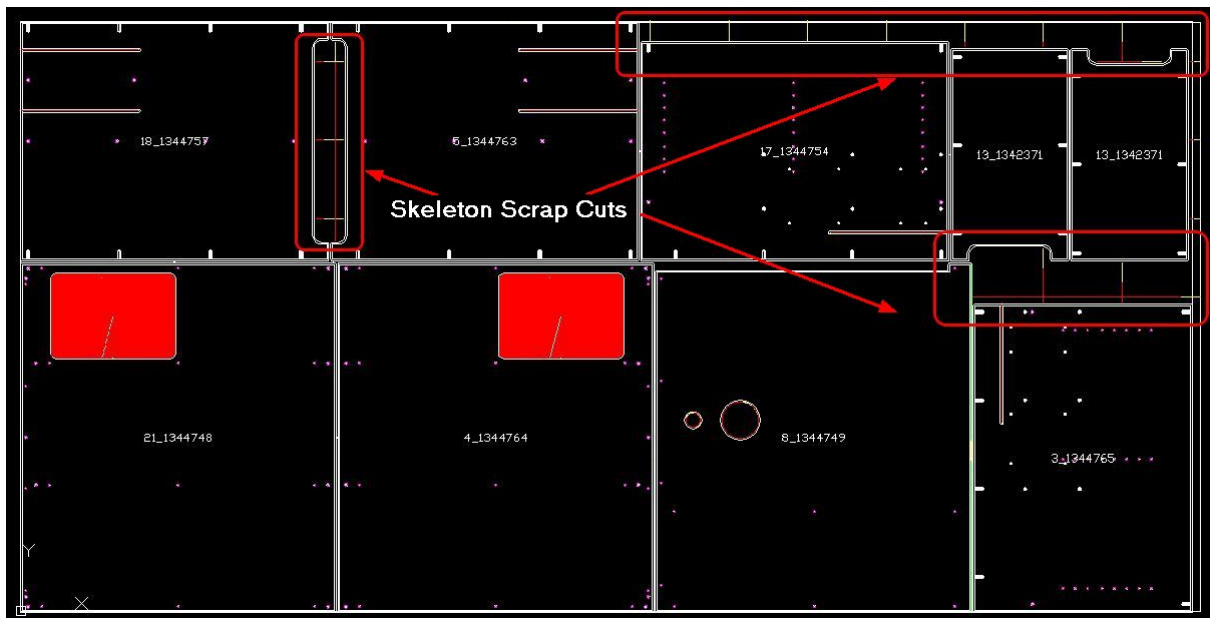
Y Spacing

Equal Spacing ☐

of X Spacings

of Y Spacings

The result of the Skeleton Scrap shown here is based on the parameters noted above:



Multi-Stock Nesting

Multi-Stock Nesting allows you to enter alternate sizes under a material in order for the nesting function to determine the best mix of materials to use.

When entering an alternate stock, you need to enter:

- YDim
- XDim
- Quantity (QTY) Note: If you enter a quantity other than 999, Router-CIM Automation Suite will only use the specified quantity till it runs out. It will then go to the next available stock size.
- Priority
- Cost (Optional)

You have three options when using alternate stocks:

- **Priority:** Priority is based on a 1 - 10 scale with 1 being the highest priority and 10 being the lowest priority. When nesting, Router-CIM Automation Suite will look at the priority given and use the sheets based on this figure.
- **Best Yield:** This allows Router-CIM Automation Suite to use any combination of the standard stock size and alternate stock size to create the nests based on the best yield that it can achieve. Priority is ignored.
- **Best Price:** This allows Router-CIM Automation Suite to use any combination of the standard stock size and alternate stock size to create the nests based on the best price. You need to have a cost entered into the standard stock size and all the alternate stock sizes. Priority is ignored.

Multi-Stock Nesting

☒ Priority
 ☐ Automatically Decrease Qty

☐ Best Yield

☐ Best Price

Alternate Sizes

+ Add Alternate Size - Delete Alternate Size

YDim	XDim	Qty	Priority	Cost
49.0000	121.0000	999	8	0.0000
61.0000	97.0000	999	10	0.0000
61.0000	121.0000	999	10	0.0000

When using Multi-Stock nesting, you can build a separate material in order to use a unique material description and material code.

First, in Automation go to File > Settings > [Advanced Settings](#) and check the box for “Export all materials during job run”. Restart Automation.

Add your materials to the material database. In this example, PLYWD_0.7500 is the main material and the other two are the alternates.

Material Database

Edit Material New Material Duplicate Material Delete Material Import/Export Delete Scrap Delete Alternate Stock Sizes

MatDesc	Attribute5	MatCode	Thickness
PLYWD_0.7500		plywdpC	0.7500
PLYWD_0.7500-5X10	plywdpC	plyxxB	0.7500
PLYWD_0.7500-5X12	plywdpC	plyxxA	0.7500

Materials With Scrap Materials with Alternate Sizes Materials in Use Materials Not in Use All Materials

Display is NOT filtered Total materials in database: 3 Total displayed materials: 3 Verify Material Database Integrity Close

On the properties of the main material, add the sizes of your alternates under the Advanced Nesting tab. These must match the sizes of the alternates entered above.

Material Properties

Material Description: PLYWD_0.7500

Material Code: plywdC Material Handling Code: ☐ Z0 is top of Material

Stock Settings

Sheet Stock	49.0000	Quantity	9999
Sheet Stock	97.0000	Priority	5
Thickness	0.7500		
Bridge Width	0.6250		
Cost	54.00		

Edge Allowances

Top: 0.1250

Left: 0.1250 Right: 0.1250

Bottom: 0.1250

Irregular Stock Edge Allowance: 0.1250

Material Rotation

☒ Allow Part Rotation (All) ☐ No Part Rotation (0)

☐ Part Rotation (0 90 180) ☐ Part Rotation (0 90)

☐ Cross Grain (90 270) ☐ Grain Rotation (0 180)

☐ Custom Rotation

Material Attributes

One:

Two:

Three:

Four:

Alt Material Code:

Scrap Management **Advanced Nesting Parameters**

Skeleton-Cut-Off Parameters

☐ Enable Skeleton-Cut-Off

Offset Distance: 0.0000

X Spacing: 0.0000

Y Spacing: 0.0000

Equal Spacing: ☐

of X Spacings: 0.0000

of Y Spacings: 0.0000

Multi-Stock Nesting

☒ Priority ☐ Best Yield ☐ Best Price

☐ Automatically Decrease Qty

Alternate Sizes

YDim	XDim	Qty	Priority	Cost	CustomID
60.0000	120.0000	999	10	66.00	
60.0000	144.0000	999	15	78.00	

Cart Control Parameters

Jobs per Cart: 100

Open Cart Threshold: 95

☒ Allow Veneer Matching

Make Default Material For Import Wizards

On the properties of the alternate sized materials, enter the material code of the main material. Do not enter anything in this field on the main material.

Material Properties

Material Description: PLYWD_0.7500-5X10

Material Code: plyxB Material Handling Code: ☐ Z0 is top of Material

Stock Settings

Sheet Stock	60.0000	Quantity	999
Sheet Stock	120.0000	Priority	5
Thickness	0.7500		
Bridge Width	0.6250		
Cost	66.00		

Edge Allowances

Top: 0.1250

Left: 0.1250 Right: 0.1250

Bottom: 0.1250

Irregular Stock Edge Allowance: 0.1250

Material Rotation

☒ Allow Part Rotation (All) ☐ No Part Rotation (0)

☐ Part Rotation (0 90 180) ☐ Part Rotation (0 90)

☐ Cross Grain (90 270) ☐ Grain Rotation (0 180)

☐ Custom Rotation

Material Attributes

One:

Two:

Three:

Four:

Alt Material Code:

Scrap Management **Advanced Nesting Parameters**

☒ Make a Scrap Cut ☐ Inventory Scrap ☐ Use Scrap

☐ Inventory to other Material

Scrap

ID	YDim	XDim	Qty	Priority	Cost	Bin	Availability
----	------	------	-----	----------	------	-----	--------------

Scrap Cut Properties

☐ Longest Side Only ☐ Shortest Side Only ☐ Longest Side First ☐ Vertical Only ☐ Horizontal Only ☐ Vertical First ☐ Horizontal First ☐ Combined

Scrap Cut Diagram

Min. X: 12.0000

Min. Y: 12.0000









Distance from Part in X: 0.6250

Distance from Part in Y: 0.6250

Scrap Cut Extension Distance: 0.1875

Make Default Material For Import Wizards

When you create a job, only assign the main material to the parts. When the job is run and an alternate size stock entered on the main material is encountered, it will use the material code assigned to the alternate size entered in the material database (plyxxA or plyxxB in this example).

Parts						
Job Settings Nesting Advanced Nesting Dynamic Variables Printing and Labels Part Name Output File						
	Part#	Part	Qty	Material		Full Path to Part
	1	BEST-island_1.d...	8	PLYWD_0.7500	▼	C:\S\BEST-island\BEST-island_1.dwg
	2	BEST-island_2.d...	16	PLYWD_0.7500	▼	C:\S\BEST-island\BEST-island_2.dwg
	3	BEST-island_4.d...	12	PLYWD_0.7500	▼	C:\S\BEST-island\BEST-island_4.dwg
	4	BEST-island_5.d...	14	PLYWD_0.7500	▼	C:\S\BEST-island\BEST-island_5.dwg
	5	BEST-island_9.d...	10	PLYWD_0.7500	▼	C:\S\BEST-island\BEST-island_9.dwg
	6	BEST-island_11....	15	PLYWD_0.7500	▼	C:\S\BEST-island\BEST-island_11.dwg
	7	BEST-island_12....	9	PLYWD_0.7500	▼	C:\S\BEST-island\BEST-island_12.dwg
	8	BEST-island_13....	14	PLYWD_0.7500	▼	C:\S\BEST-island\BEST-island_13.dwg

Cart Control Parameters

Cart Control Nesting will allow you to control, to some degree, the jobs that are nested together on your sheet stock when multiple jobs are run together in Router-CIM Automation Suite. Normally if you place parts from multiple jobs together in Router-CIM Automation Suite, there is no control over which parts will end up together on a sheet, since yield is the highest concern at that point. This can lead to confusion or lost time as the parts are sorted at the machine, after being cut, so that they can be stacked together with other parts from the same job.

In the material, you can choose the number of Jobs per cart. To locate this option, go to your Material Database and select the material you want to setup for Cart Control. When you open the material, select on the Advanced Nesting Parameters tab and the Cart Control Parameters will be on the right side. This is really the number of groups of parts that have the same record description per sheet in this material. The Open cart threshold is an override that says if there is at least the percentage shown of available space then go ahead and add another group of parts with the same description on this sheet. In this case, only two carts can be on the same sheet because there would have to be 95% of the space (empty sheet) left over in order to add a third cart to the sheet.

For more information refer to the [Cart Control](#) section.

Cart Control Parameters

Jobs per Cart

2

Open Cart Threshold

95

Veneer Matching Parameters

A veneer matched set/group is the combination of multiple parts that you want to be nested together in a set configuration so that they are machined from one section of a sheet.

Check the box for 'Allow Veneer Matching' in order to use the Veneer Matching feature.

For more information refer to the [Veneer Matching](#) section.

☒ Allow Veneer Matching

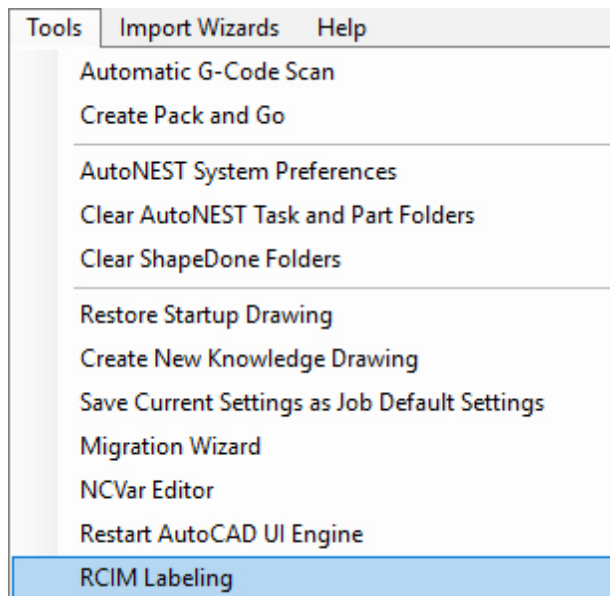
Labeling

Router-CIM Automation Suite provides label information in several formats. You can modify the labels as necessary to match your requirements.

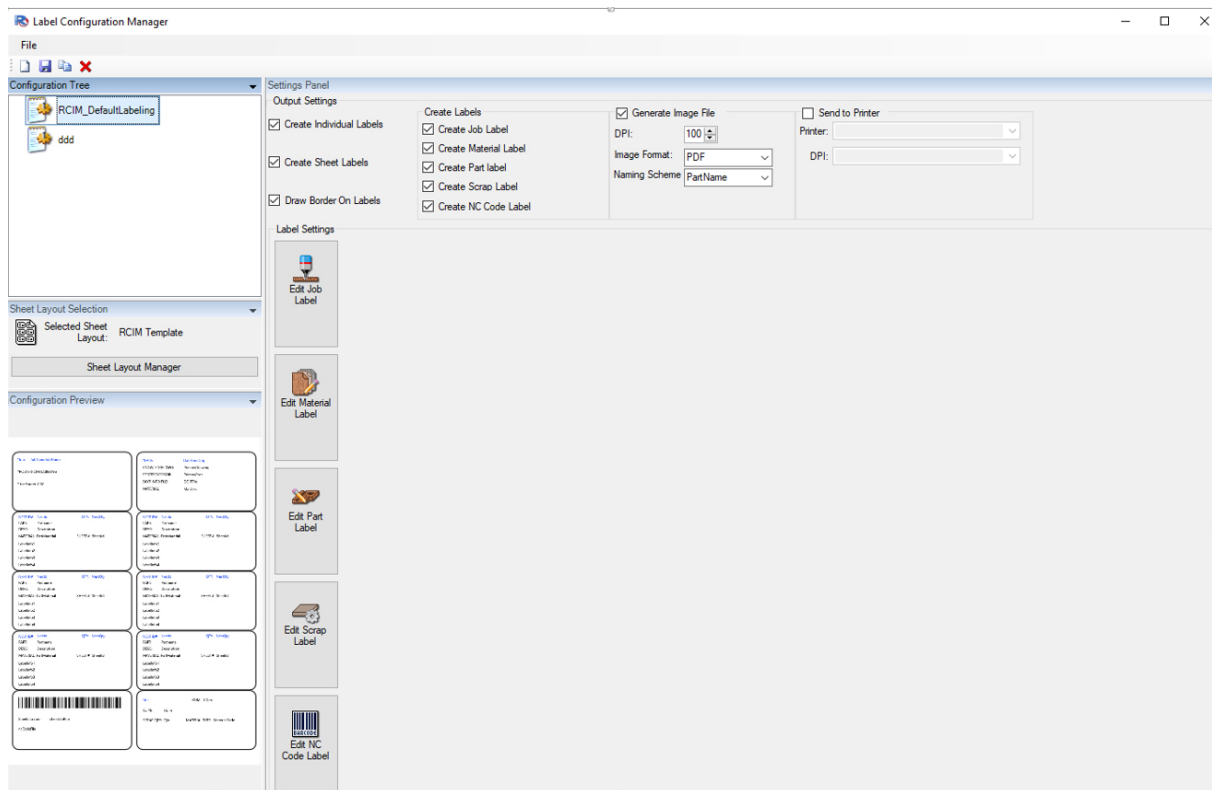
Router-CIM's Label Designer is the default labeling program. For more information go to the [Router-CIM Label Designer](#) section.

Label Designer

The Label Designer is accessed through the 'Tools' drop down in Router-CIM Automation Suite. This Label Designer allows you to make a template of where you want part properties to appear on a printed label.



The Label Designer consists of 3 areas, the Menu Bar, Sheet Layout Manager and the Label Designer.



The **Menu Bar** lets you create a new configuration, save, copy, delete, import and export label configurations.



New: This will allow you to create a new label configuration.



Save: Saves the currently selected label configuration.



Copy: Makes a copy of the currently selected label configuration



Delete: Removes the currently selected label configuration



Import: Allows you to import a label configuration from another Router-CIM Automation Suite computer



Export: Allows you to export for the currently selected label configuration for Import onto another Router-CIM Automation Suite computer

The **Sheet Layout Designer** lets you create a configuration for sheet labels that can be printed to your office laser or inkjet printer and size a label for a roll label printing system. The sheet layout will be determined by the labels that you intend to use.

To create a Sheet Layout, go to the ['Creating a Sheet Layout'](#) section.

The **Label Designer** lets you create custom labels that can be individually created or designed to match a sheet template that you have created or to create individual labels for use with the Touch-N-Print labeling system or a stand-alone printing system.

Note: If creating labels for a sheet template, the size of the label must match exactly to the label size determined by the Sheet Layout.

There are 5 different labels that you have access to:

- 1) [Part Label](#)
- 2) [Job Label](#)
- 3) [Material Label](#)
- 4) [Scrap Label](#)
- 5) [NC Code Label](#)

Select a label listed above for more information.

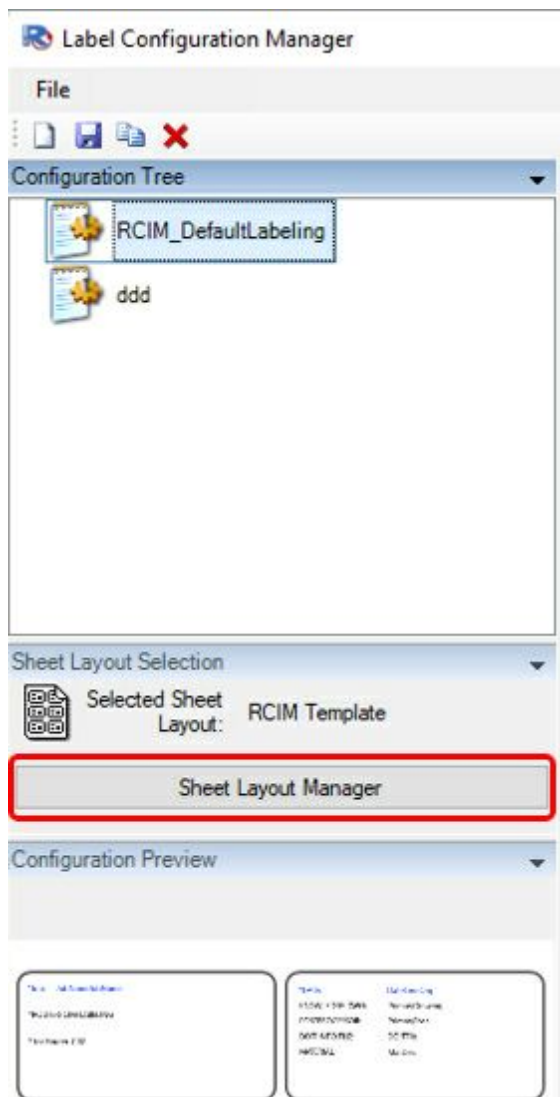
Once your labels have been designed and/or your sheet layout templates, you will create a configuration or multiple configurations that you want for your labels when a job is run in Router-CIM Automation Suite.

For information setting up a configuration for use in the job, go to the ['Automation Label Settings'](#) section.

Create a Sheet Layout Design

The Sheet Layout Manager allows the user to design how they want the labels to be output on a sheet or bitmap.

To create a Sheet Layout, open the Sheet Layout Manager from the RCIM Labeling Interface:



Note: If creating labels for a sheet template, the size of the label must match exactly to the label size determined by the Sheet Layout.

The designer has two main controls: The Layout Template selection and the template preview.

The template selection window allows the user to see existing templates and to edit them.

New Button- launches the create template window

Copy Button – allows you to copy an existing configuration

Delete button – allows you to delete an existing configuration

The template preview shows what the template looks like when the user selects it.

The screenshot shows the 'Sheet Layout Manager' dialog box. At the top, there's a 'Select Sheet Layout' dropdown menu set to 'RCIM Template'. Below it are 'New', 'Copy', and 'Delete' buttons. The 'Settings' section includes a 'Name' field with 'RCIM Template' and an 'Orientation' section with 'Portrait' selected. The 'Template Designer' section has two columns: 'Horizontal' and 'Vertical'. Each column has five rows of input fields: 'Panels', 'Page Size: (in)', 'Page Margin: (in)', 'Panel Size: (in)', and 'Panel Margin: (in)'. The 'Horizontal' column values are 2, 8.500, 0.156, 4.000, and 0.188. The 'Vertical' column values are 5, 11.000, 0.500, 2.000, and 0.000. Below these is a 'Border' section with a 'Shape' dropdown set to 'Rounded' and a 'Rounded: (inch)' field set to 0.00. To the right is a 'Template Preview' section showing a 2x5 grid of rounded rectangles. A 'Close' button is at the bottom right.

	Horizontal:	Vertical:
Panels:	2	5
Page Size: (in)	8.500	11.000
Page Margin: (in)	0.156	0.500
Panel Size: (in)	4.000	2.000
Panel Margin: (in)	0.188	0.000

Border:
Shape: Rounded
Rounded: (inch) 0.00

Template Preview: (2x5 grid of rounded rectangles)

Create a New Sheet Layout

To create a new sheet layout, select the 'New' button.

Sheet Layout Manager

Select Sheet Layout: New Layout

Buttons: New, Copy, Delete

Settings:

Name: New Layout

Orientation: ☒ Portrait ☐ Landscape

Template Designer:

	Horizontal:	Vertical:
Panels:	1	1
Page Size: (in)	8.500	11.000
Page Margin: (in)	0.000	0.000
Panel Size: (in)	8.500	11.000
Panel Margin: (in)	0.000	0.000

Border:

Shape: Rounded

Rounded: (inch) 0.00

Template Preview:

Close

Name: Name of the template

Orientation: The orientation of the template

Panels: The number of labels in horizontally and vertically on the sheet.

Page Size: This is the size of the sheet in inches

Page Margin: this is the distance horizontal and vertical from the edge of the sheet the labels will start

Panel Size: This is the size of the panel in inches. It should match what the user designs as a label template

Panel Margin: This is the distance from one panel to another

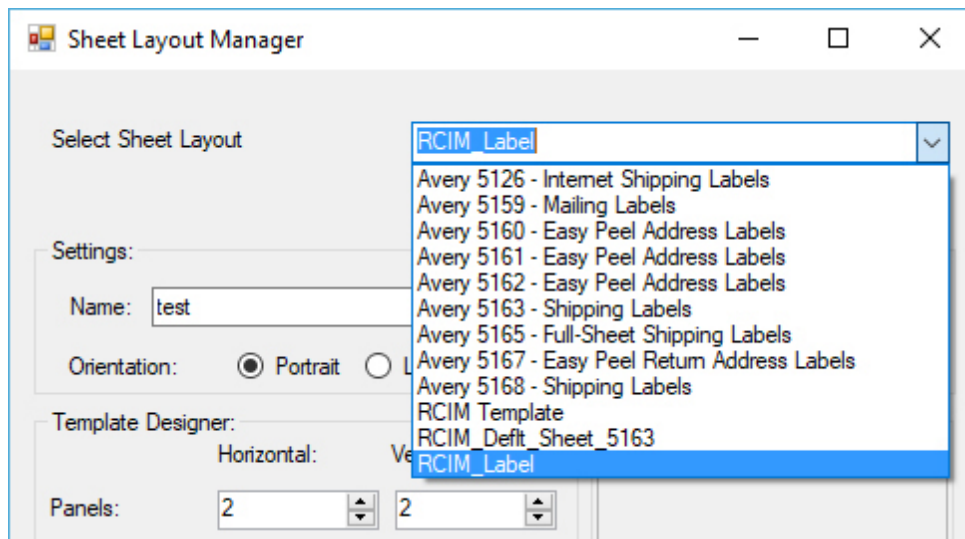
Note: Each field needs to be completed in order to make sure that when the labels are printed, that they match to the sheet labels that you intend to use.

Once you have completed setting up the sheet layout, your Sheet Layout name will automatically show up in the drop-down window by 'Select Sheet Layout' at the top of the Sheet Layout Manager.

Your Sheet Layout is now ready to be used in a [Router-CIM Label Configuration](#).

Edit an Existing Sheet Layout

To edit an existing sheet layout, simply select the sheet layout from the drop-down window by 'Select Sheet Layout' at the top of the Sheet Layout Manager.



Once you selected the sheet layout, make the changes you need and the process will automatically update the sheet layout.

Copy an Existing Sheet Layout

To copy an existing sheet layout, simply select the sheet layout from the drop-down window by 'Select Sheet Layout' at the top of the Sheet Layout Manager.

Once you selected the sheet layout, select the 'Copy' button.

This will make a copy of the sheet layout by adding _Copy to the end of the sheet layout name. You can adjust the name at this time and it will update in the list.

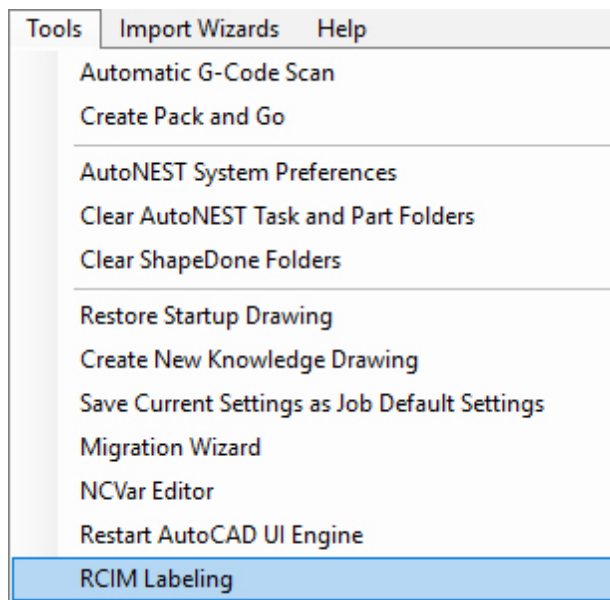
Delete an Existing Sheet Layout

To delete an existing sheet layout, simply select the sheet layout from the drop-down window by 'Select Sheet Layout' at the top of the Sheet Layout Manager and select the 'Delete' button.

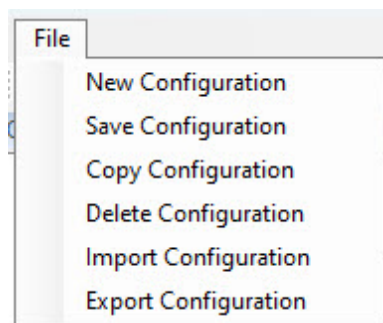
Creating a Label

Creating a custom label using the Router-CIM Label Designer is a simple process:

Open RCIM Labeling. The RCIM Labeling is accessed through the 'Tools' drop down in Router-CIM Automation Suite.



Once the Router-CIM Label Designer opens, go to the upper left and select the 'File' menu and the option 'New'. This will create a new configuration:



New Configuration - Creates a new label configuration for the user to design

Save Configuration - Saves the selected configuration

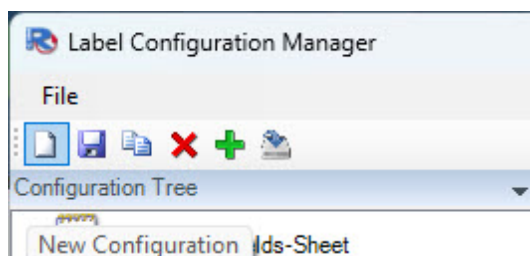
Copy Configuration – Copies the selected configuration

Delete Configuration – Deletes the selected configuration

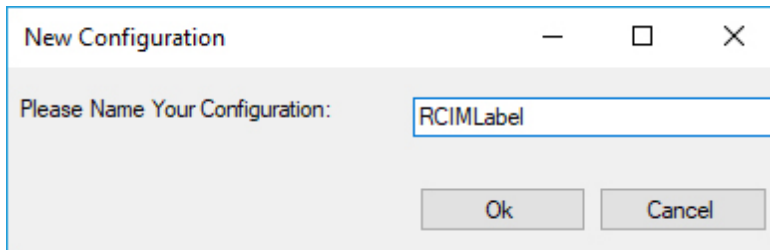
Import Configuration – Allows you to import a label configuration from another Router-CIM Automation Suite computer

Export Configuration – Allows you to export for the currently selected label configuration for Import onto another Router-CIM Automation Suite computer

Or you can select the new option

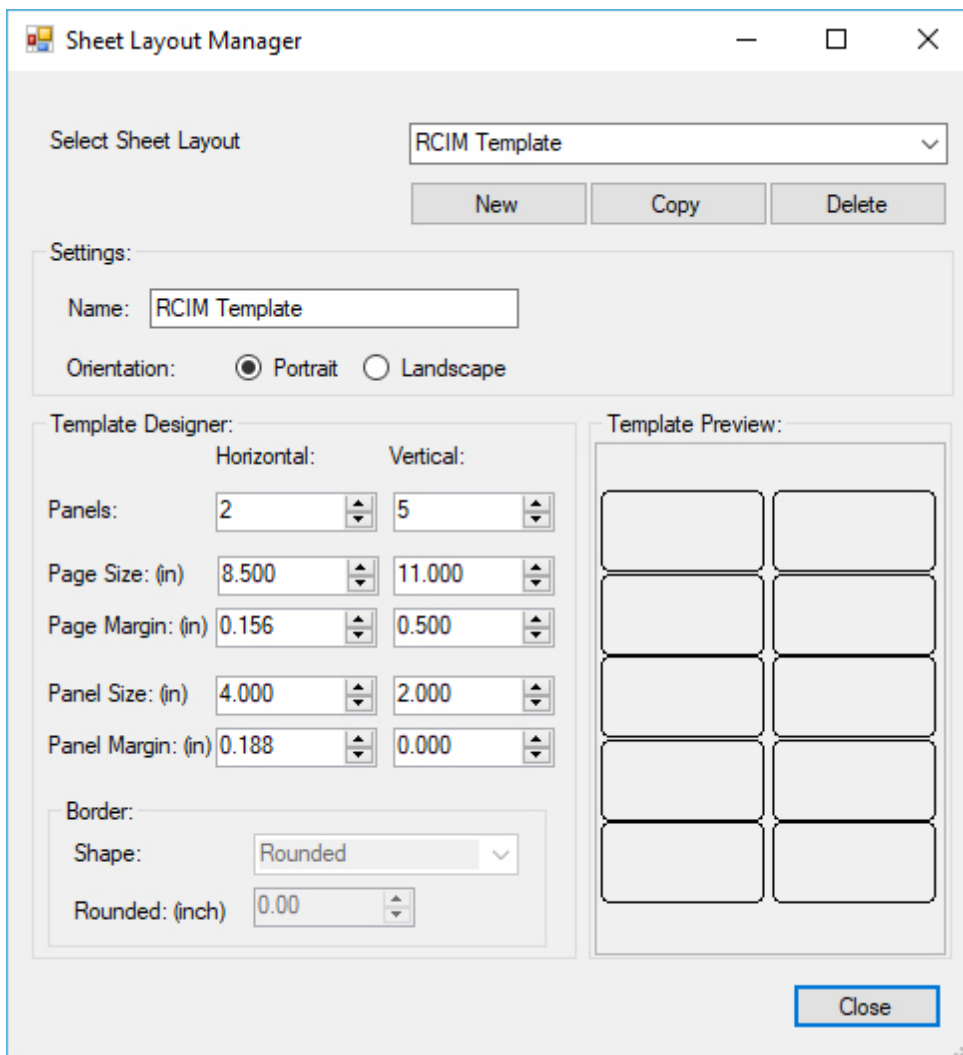


Once 'New Configuration' is selected, you will be able to name your label configuration:

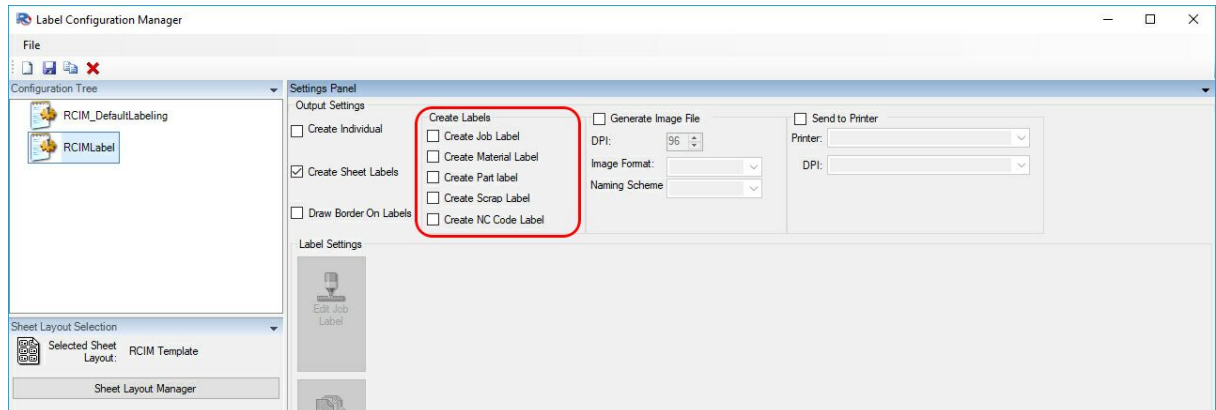


After naming your configuration, you will be directed to select a Sheet Layout. At this point, you can create a new sheet layout or select from an existing sheet layout.

For more information on creating a new sheet layout, please go to the ['Creating A Sheet Layout'](#) section.



Once a sheet layout is selected, you will then be able to select the type of labels you would like to create with this configuration.

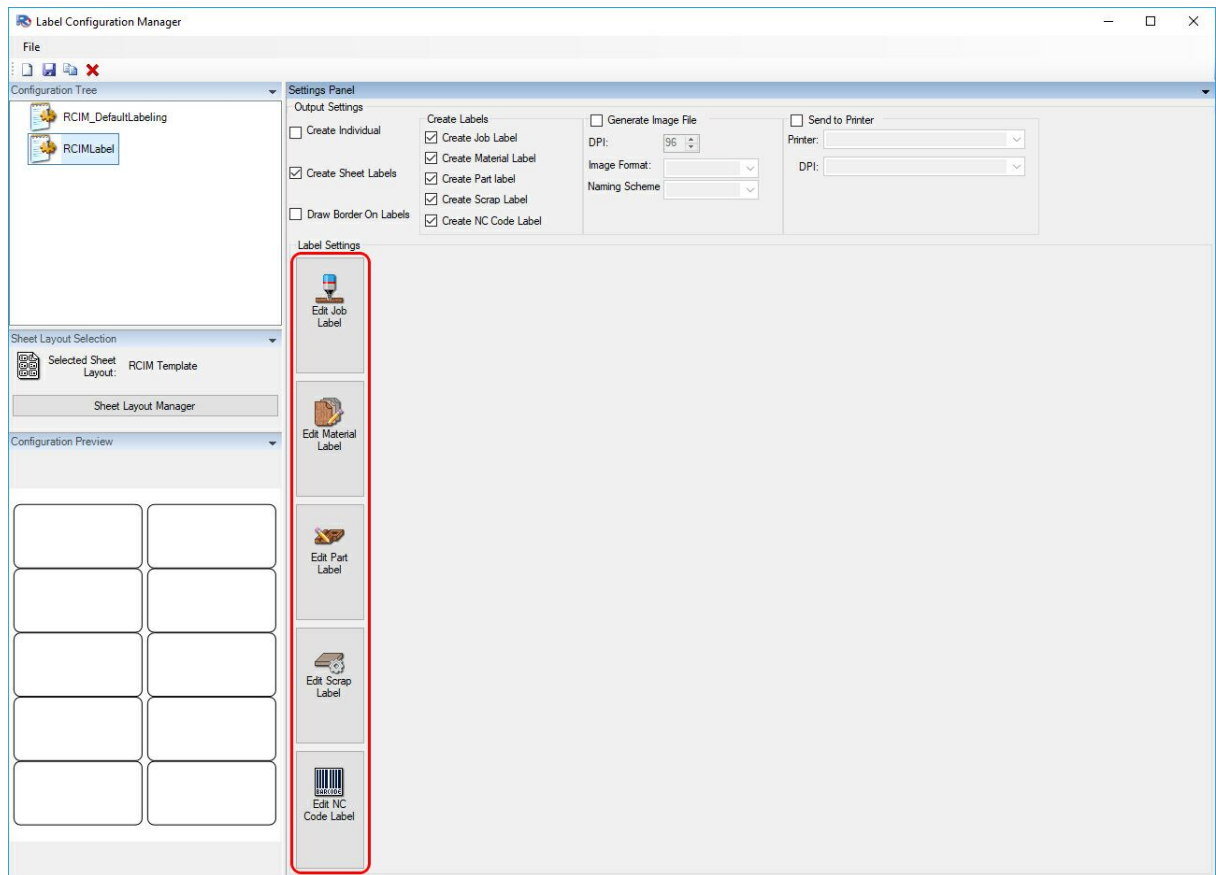


Each label configuration can have up to 5 label types:

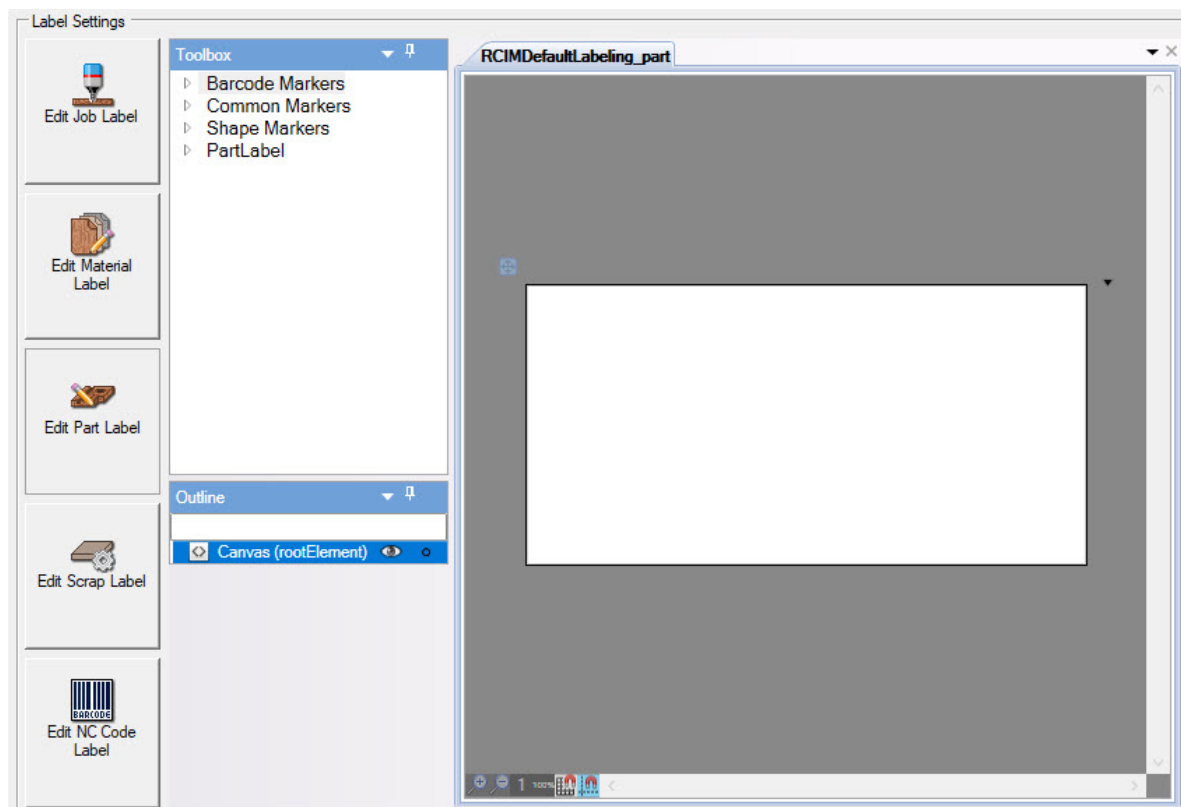
- Job Label
- Material Label
- Part Label
- Scrap Label
- NC Code Label

Each label type corresponds to the available fields accessible once the label has been created. The available fields for each label are located at the end of this section.

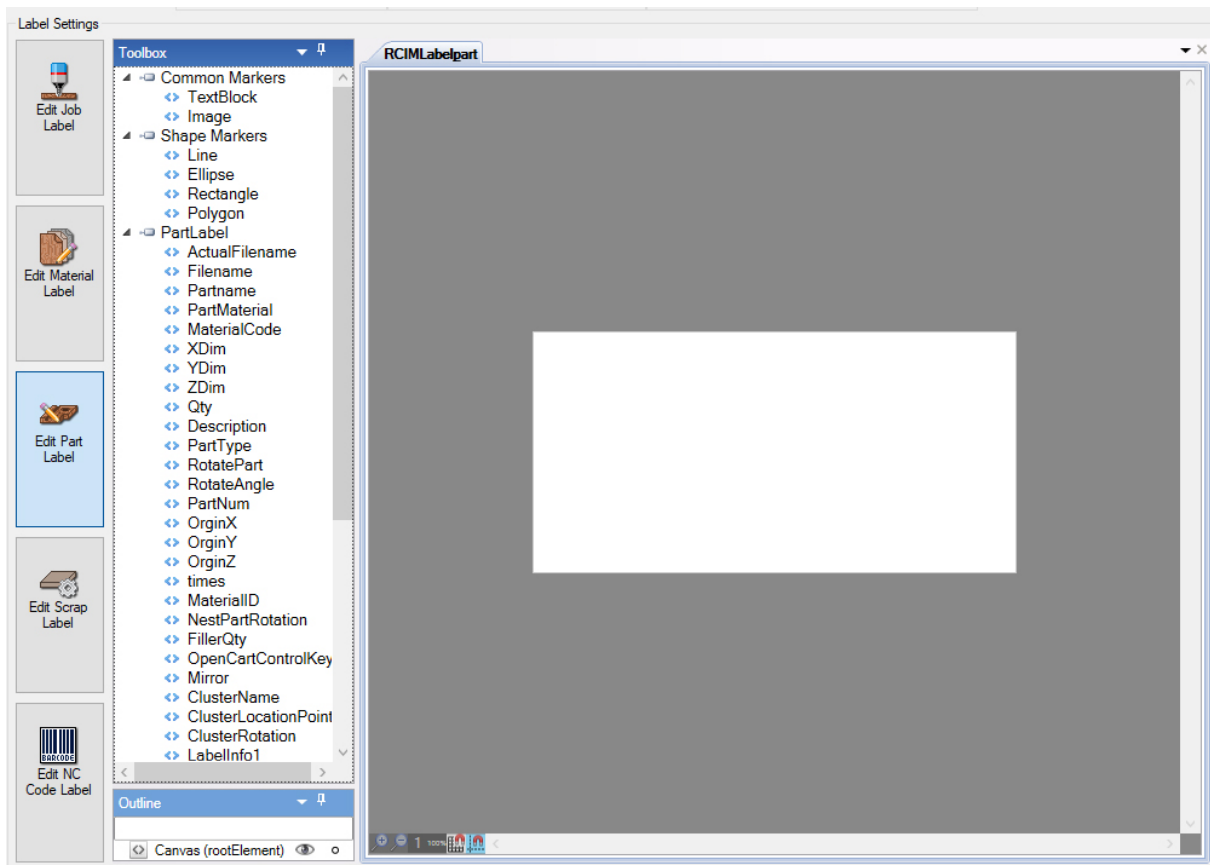
When the label types are selected, you will have access to the edit label options below. You will select the label type you want to edit and this will load into the Label Settings window.



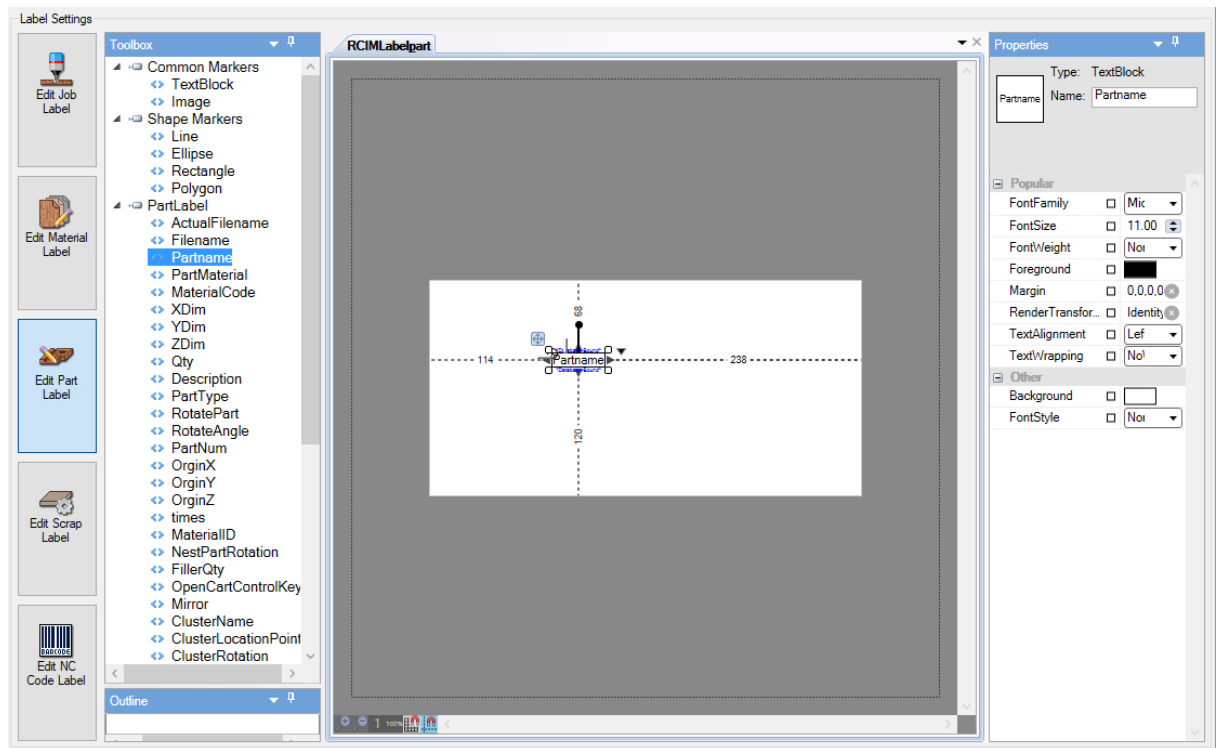
Once a label type is selected, you can start to create that label:



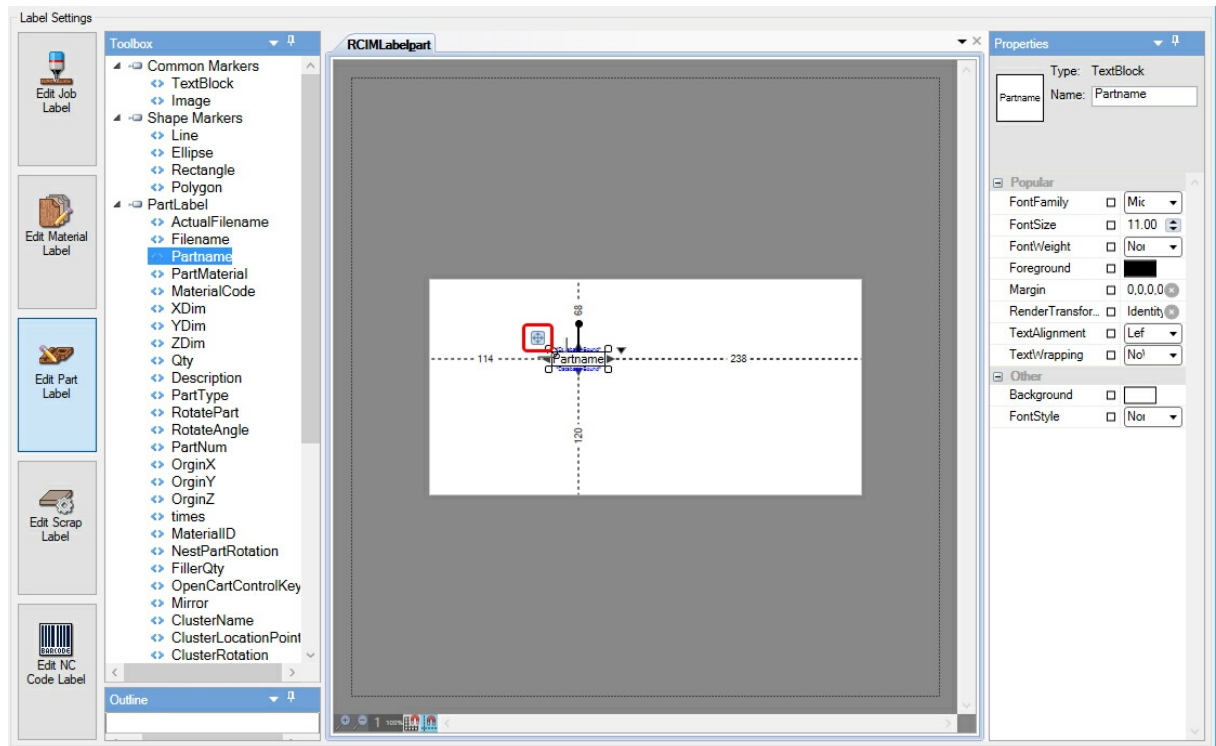
The available fields for the label are located on the left side of the Label Designer. Expand each to show the available fields for that type of label:



To add a field to you label, simply select the field you want and then click in the label to place the field on the label or you can select the field you want and drag and drop it onto the label:



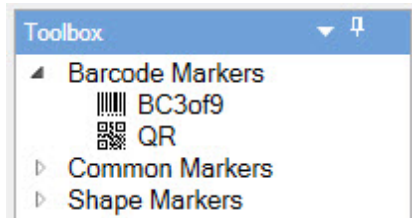
To move the field, hold down on the cross hairs to drag the field to the desired location on the label:



Repeat the process to add all the label fields that you need to your label.

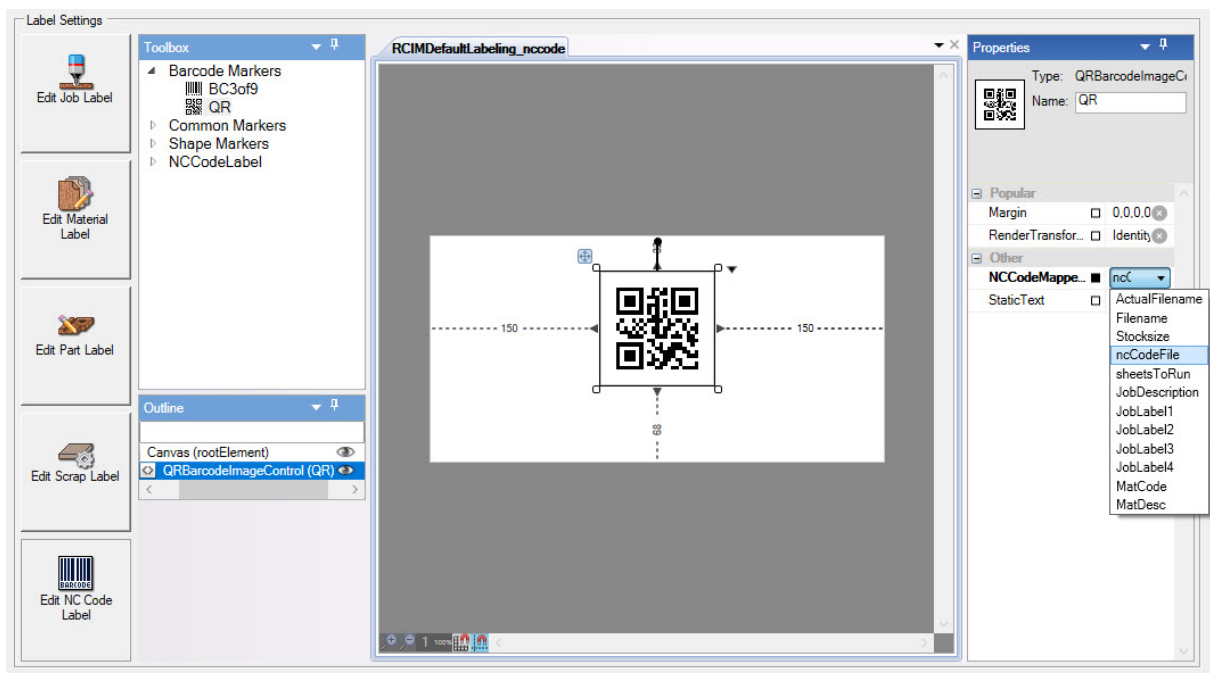
Barcodes and/or QR Codes

To add a barcode or QR code, expand the 'Barcode Markers' section.

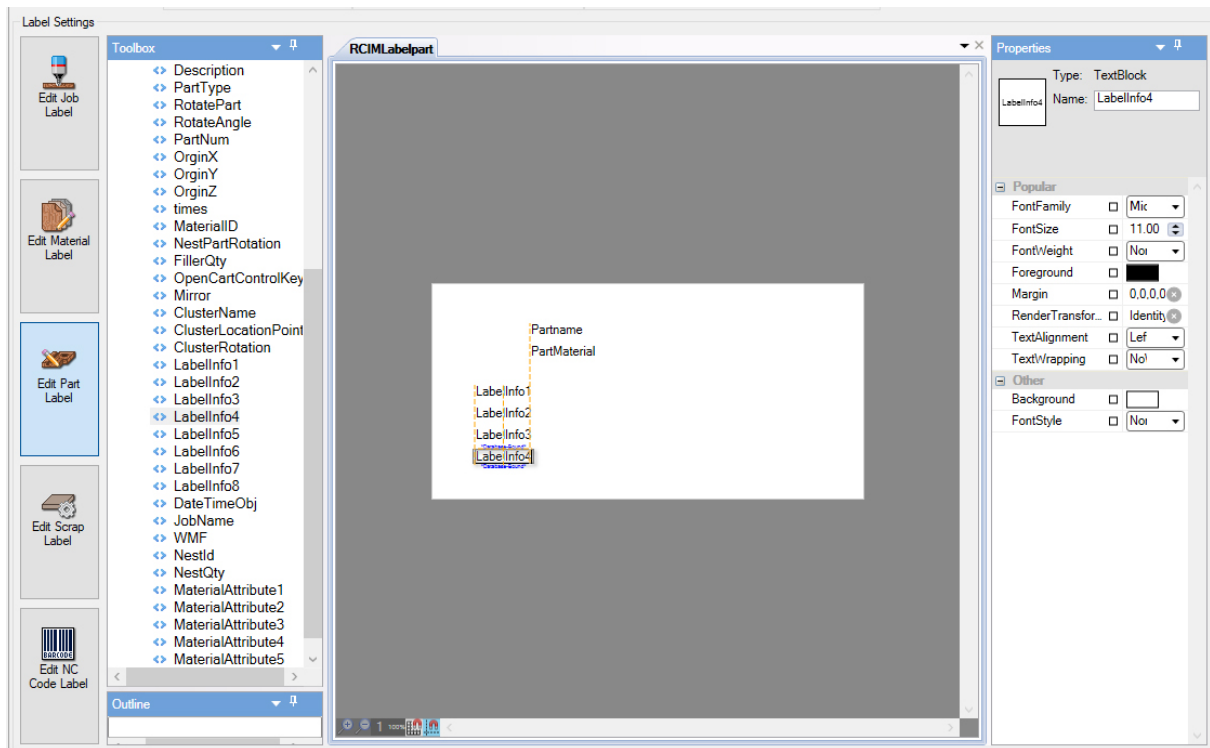


Just like adding any other field, simply select the BC3of9 or QR and place it where you would want the field on the label.

When you have it placed, on the right side under 'Properties', select the drop-down for the NCCodeMapped option to select the available fields for the Barcode or QR code.

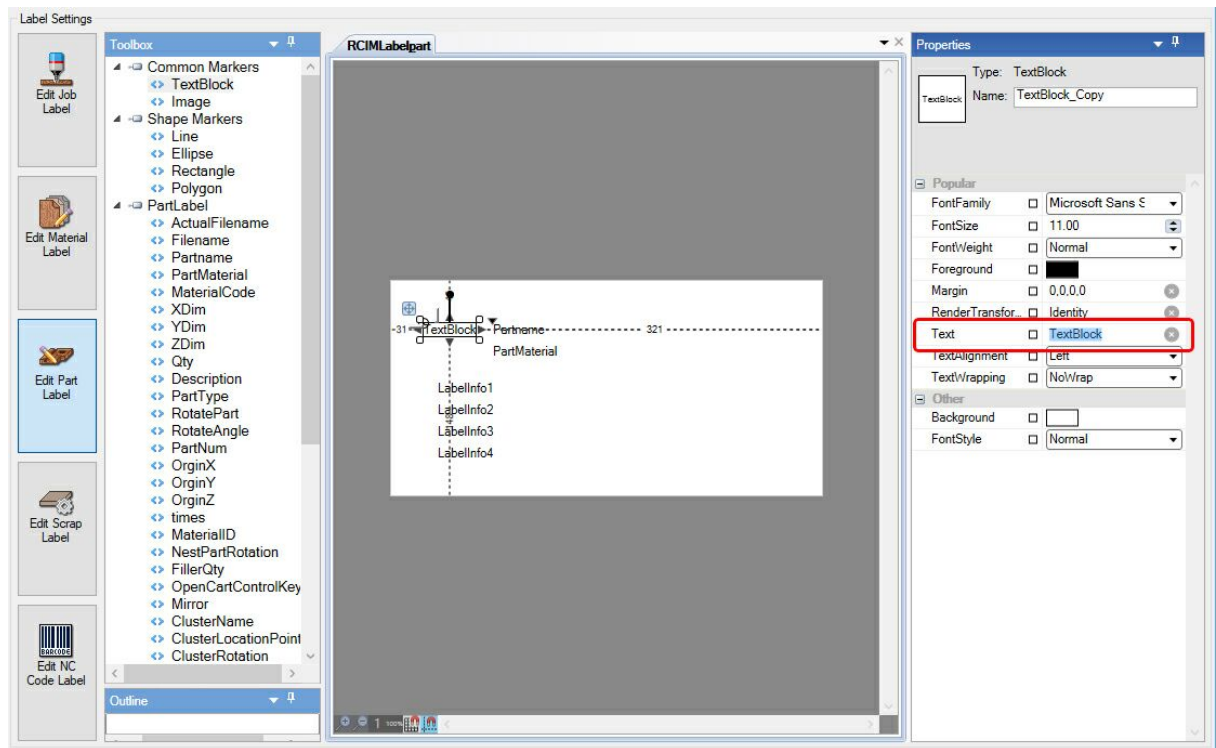


Once your fields have been added, you can use the alignment options by right clicking on a field or move the fields to the correct locations and follow the auto-alignment feature:

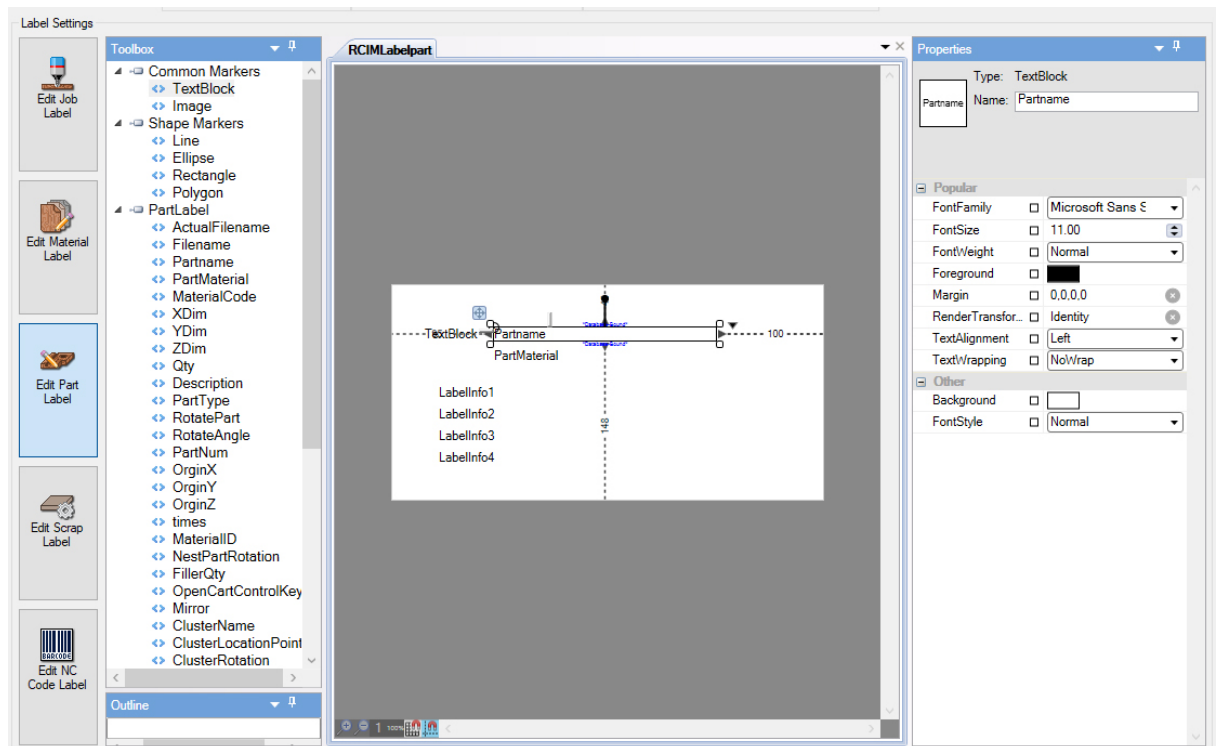


The label fields will display the information provided by Router-CIM Automation Suite. A text block can be added as an identifier by using the TextBlock field under Common Markers. This is also where you can add an image such as a company logo.

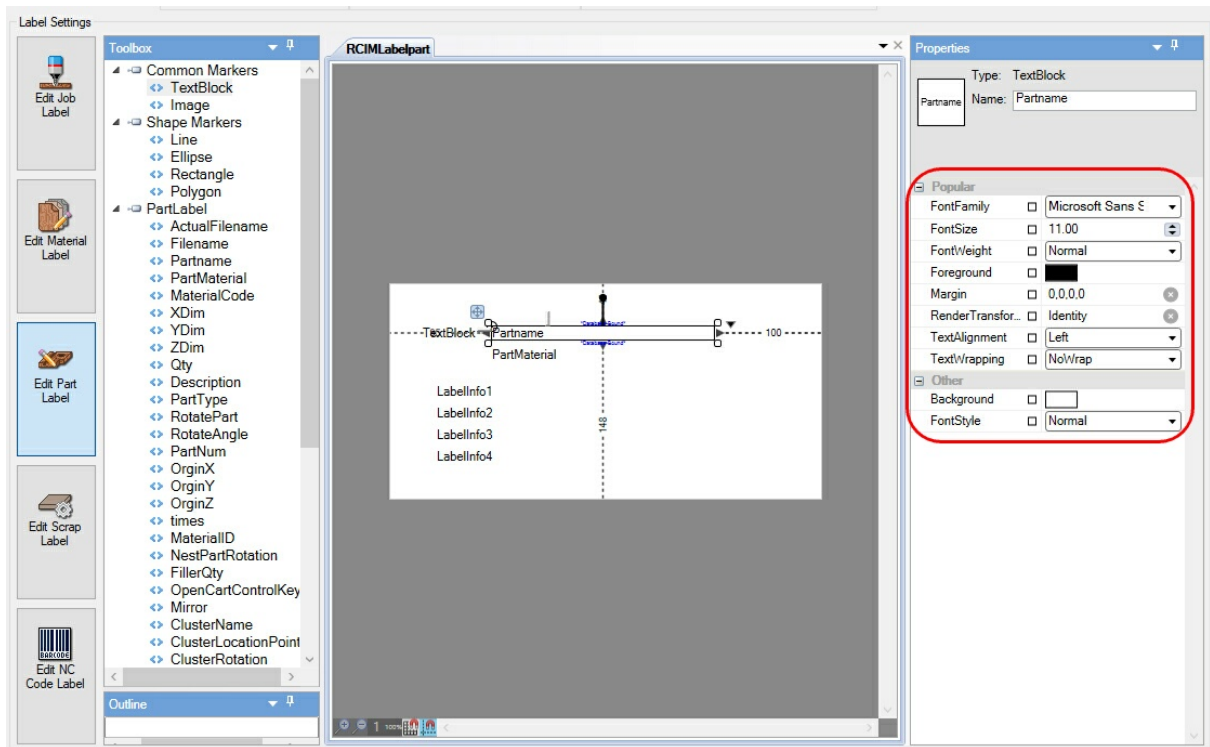
To change the text, you can select in the TextBlock field or adjust the text located under Properties on the right side of the default label screen.



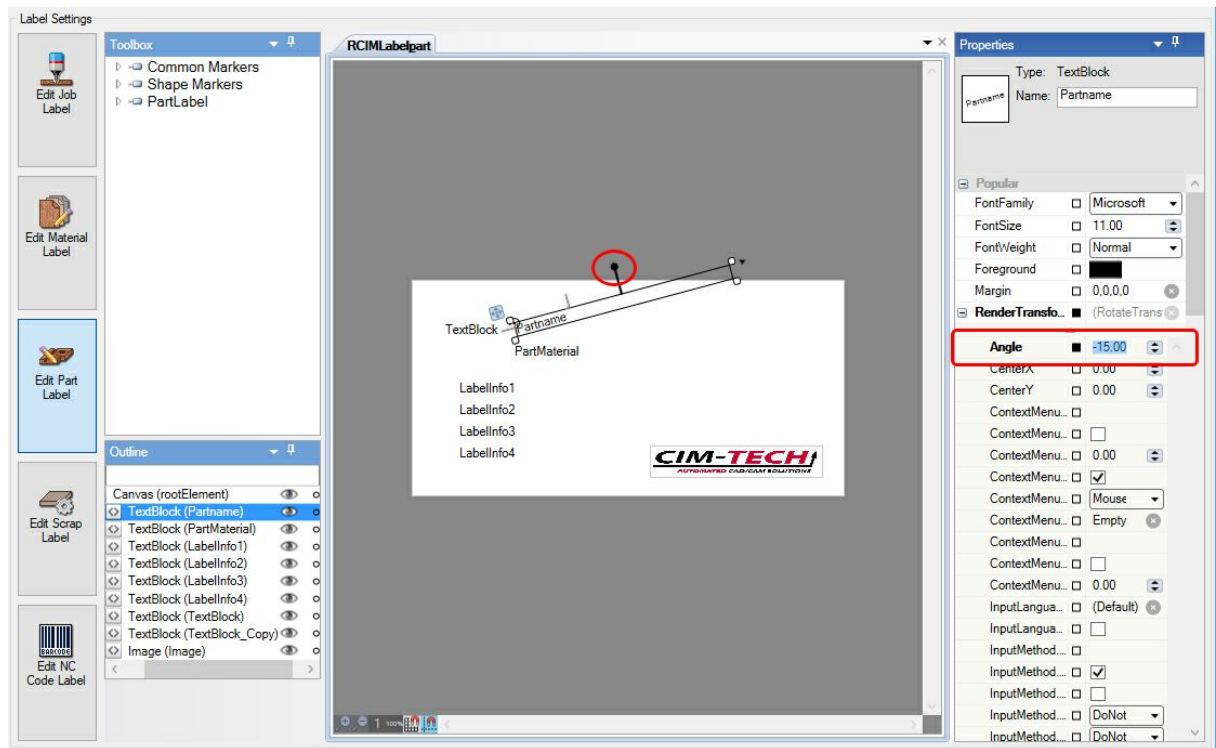
All fields can be sized to match the information you will be providing. To size a field, use the arrows to adjust length and width:



When a field is selected, on the right side of the Label Designer are the properties for that particular field. You can adjust items such as color, font size, font style, font family, text alignment and wrapping:

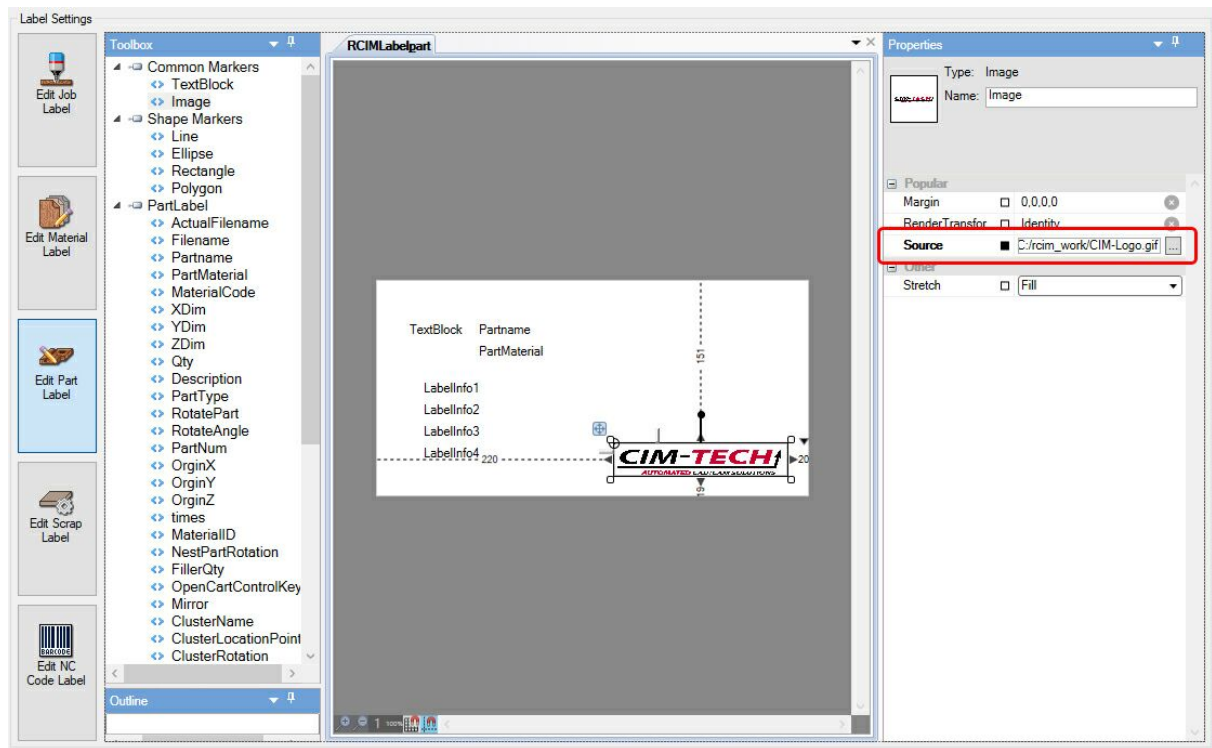


To adjust the rotation of the field, select the field that you would like to rotate and grab the rotate grip as indicated below. While holding the left mouse button down, you can move left or right to adjust the rotation angle of the field. For more precise rotation angles once you have started, on the right side you will see a heading 'Render Transform'. Select the + symbol to expand the options. Directly underneath the heading you will see the Angle field. You can type in the precise angle that you would like the field to be rotated.



Under Common Markers, you can also insert an image such as a company logo. Under the properties for the image field, you will see the 'Source' option. This is where you define the file path for the location of the image. Use the gray box with the three dots to navigate to the image file location. Acceptable file types are JPEG, PNG, JPG or GIF.

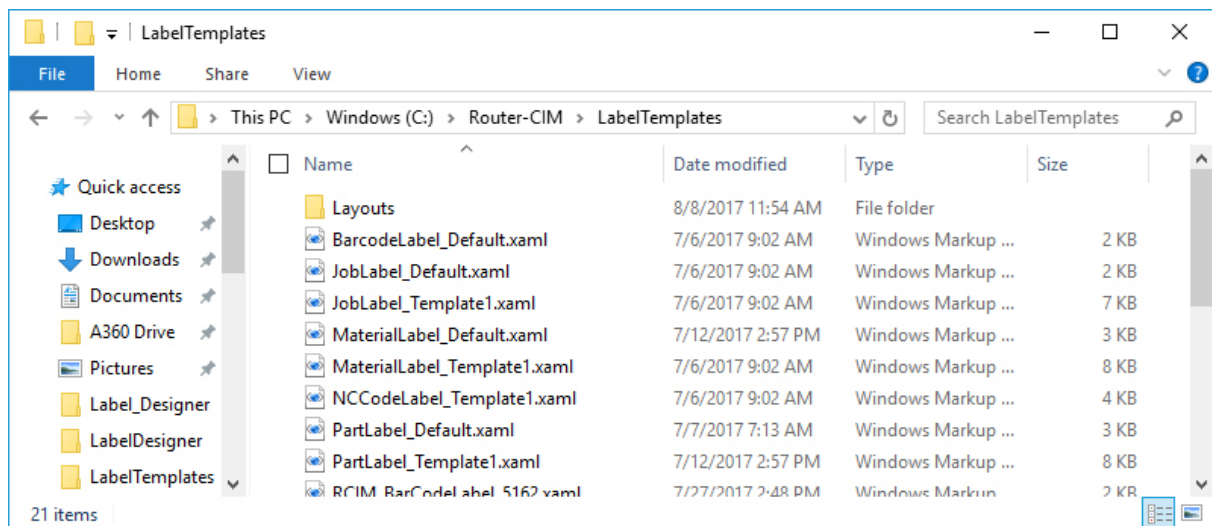
Note: This is NOT the image of the part in the part label. Refer to the [Part Label Fields Available](#) section below for information on placing a part preview picture on the labels.



Once you have all the fields, text blocks and/or images you need for the label, it is then time to save the label.

Go to the upper left of the Label Configuration Manager and select 'File' and then 'Save' or select the save icon.

Labels will be saved in this location: **C:\Router-CIMLabelTemplates**



Repeat the above steps for each required label if necessary.

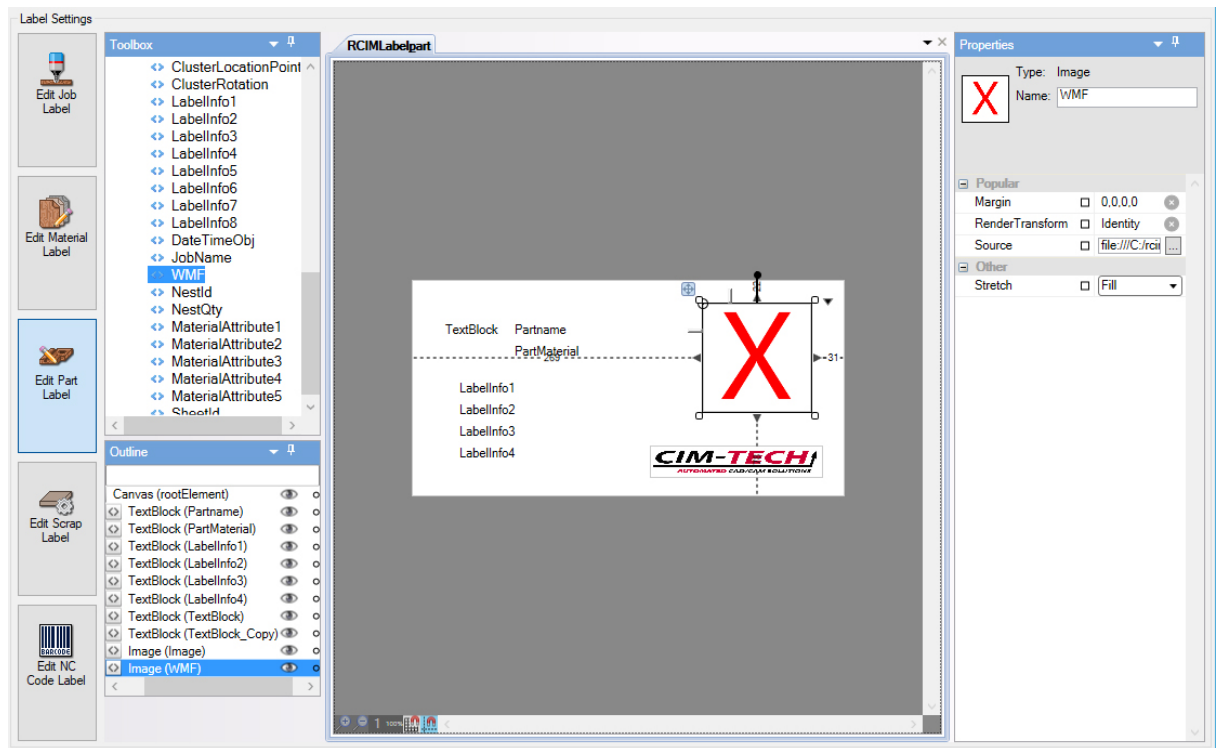
Your label is now ready to be used in a [Router-CIM Label Configuration](#).

Part Label Fields Available:

Part Name	Job Name
Part Material	Job Description
Material Code	Part Type (DWG, DXF or SCN)
Material ID	Veneer Match Name
RCIM Part Number	Veneer Match Location Point
XDIM	Veneer Match Rotation
YDIM	Material Attribute 1
ZDIM	Material Attribute 2
Quantity	Material Attribute 3
Filler Quantity	Material Attribute 4
Description	Material Attribute 5
Label Field 1	Origin Shift X
Label Field 2	Origin Shift Y
Label Field 3	Origin Shift Z
Label Field 4	Rotate Part (True or False)
Label Field 5	Rotated Angle
Label Field 6	Part Mirrored (True or False)
Label Field 7	Cart Control Key
Label Field 8	Nest Part Rotation
Label Field 9	Origin X
Label Field 10	Origin Y
Label Field 11	Origin Z
Label Field 12	Job Label 1
Label Field 13	Job Label 2
Label Field 14	Job Label 3
Label Field 15	Job Label 4
Label Field 16	Part Multiplier
Date and Time	Code Single Code File
Nest ID	Code Single Code File Full Path
Sheet ID	
Nested Quantity	

You can also add a part preview of the image to the label. In order to add a part preview use a Part label, insert the field from the left hand side called 'WMF'. Once added, you can move and size the image box to your desired look. Router-CIM Automation Suite will use this field to place a picture of the part that was processed:

Note: If you would like to have the part preview on the label, make sure that 'Place Preview on Labels' is selected under the '[Label Generation Settings](#)'.



Job Label Fields Available:

Date and Time
 Job Name
 Router-CIM Version
 Job Description
 Job Label 1
 Job Label 2
 Job Label 3
 Job Label 4

Material Label Fields Available:

Date and Time
 Job Name
 Nest Name
 Material Description
 Material Code
 Thickness
 Material Attribute 1
 Material Attribute 2
 Material Attribute 3
 Material Attribute 4
 Material Attribute 5
 Job Label 1
 Job Label 2
 Job Label 3
 Job Label 4
 Material X Dimension

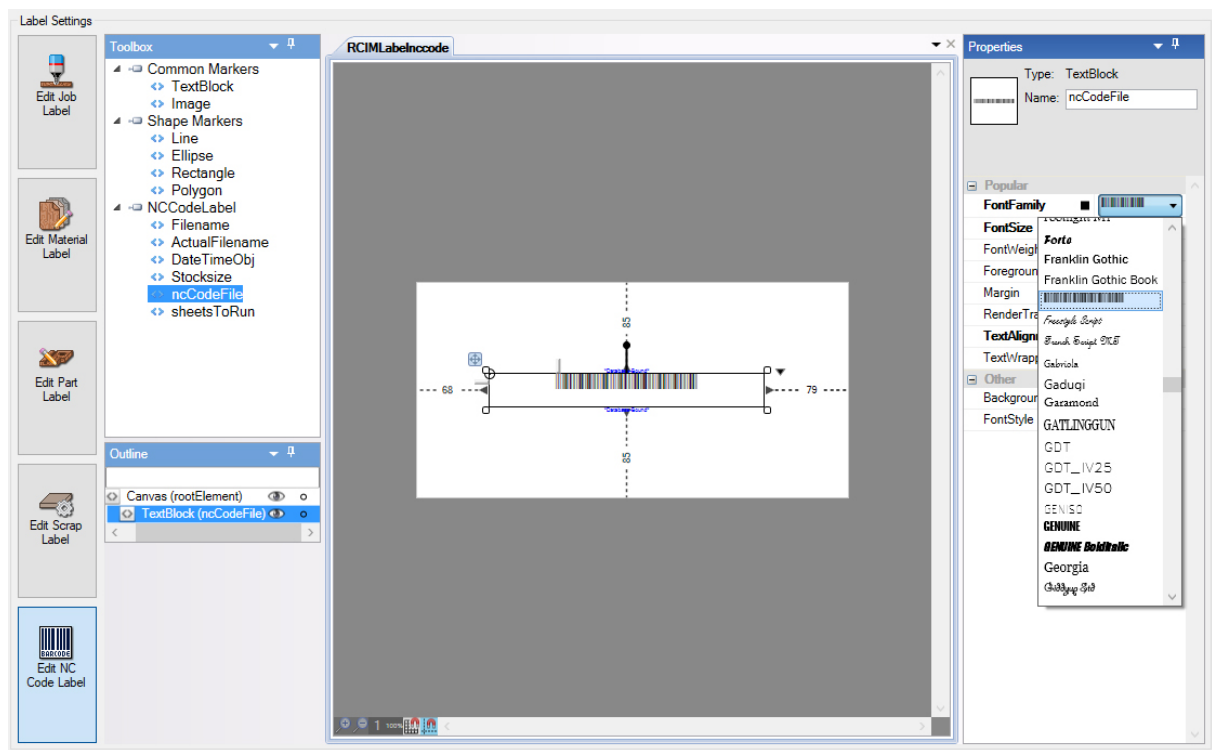
Material Y Dimension
Primary Knowledge Drawing
Primary Post Processor
Primary DOIT File

Scrap Label Fields Available:

Date and Time
Material Description
Material Code
Scrap ID
Thickness
X Dimension
Y Dimension
Job Label 1
Job Label 2
Job Label 3
Job Label 4
Bin
Material Cost

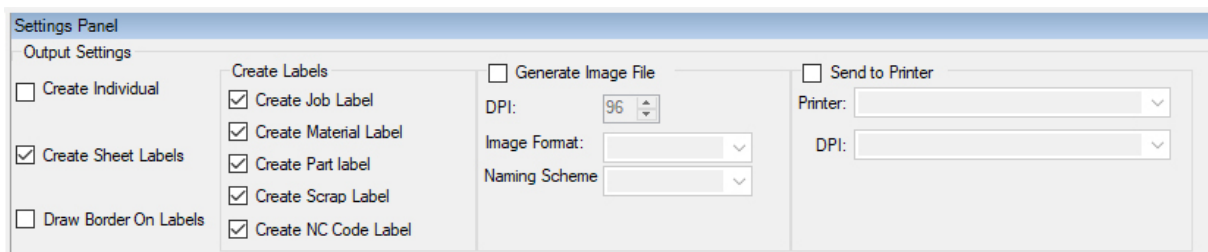
NC Code Label Fields Available:

Date and Time
NC Code File Name
Job Name
Number of Sheets to Run
Stock Size



Define Label Outputs

Once a Sheet Layout and the labels have been created, you will need to define the output files that you are looking for.



There are 2 types of outputs you can use:

- 1) Create Individual Labels
- 2) Create Sheet Labels (Requires a [Sheet Layout](#))

Create Individual Labels

Create Individual Labels will produce image files and/or be sent directly to a printer.

The screenshot shows the 'Settings Panel' with the 'Output Settings' tab selected. In the 'Create Labels' section, the 'Create Individual' checkbox is highlighted with a red rectangle. Other checkboxes in this section include 'Create Sheet Labels', 'Draw Border On Labels', 'Create Job Label', 'Create Material Label', 'Create Part label', 'Create Scrap Label', and 'Create NC Code Label'. In the 'Generate Image File' section, the 'Generate Image File' checkbox is checked, and the 'Image Format' is set to 'PDF'. The 'Naming Scheme' is set to 'PartName'. The 'Send to Printer' section is currently unchecked.

Create Sheet Labels

Create Sheet Labels will produce image files and/or be sent directly to a printer

This screenshot is similar to the previous one, but the 'Create Sheet Labels' checkbox in the 'Create Labels' section is highlighted with a red rectangle. The 'Generate Image File' checkbox remains checked, and the 'Image Format' is still 'PDF'.

Generate Image File

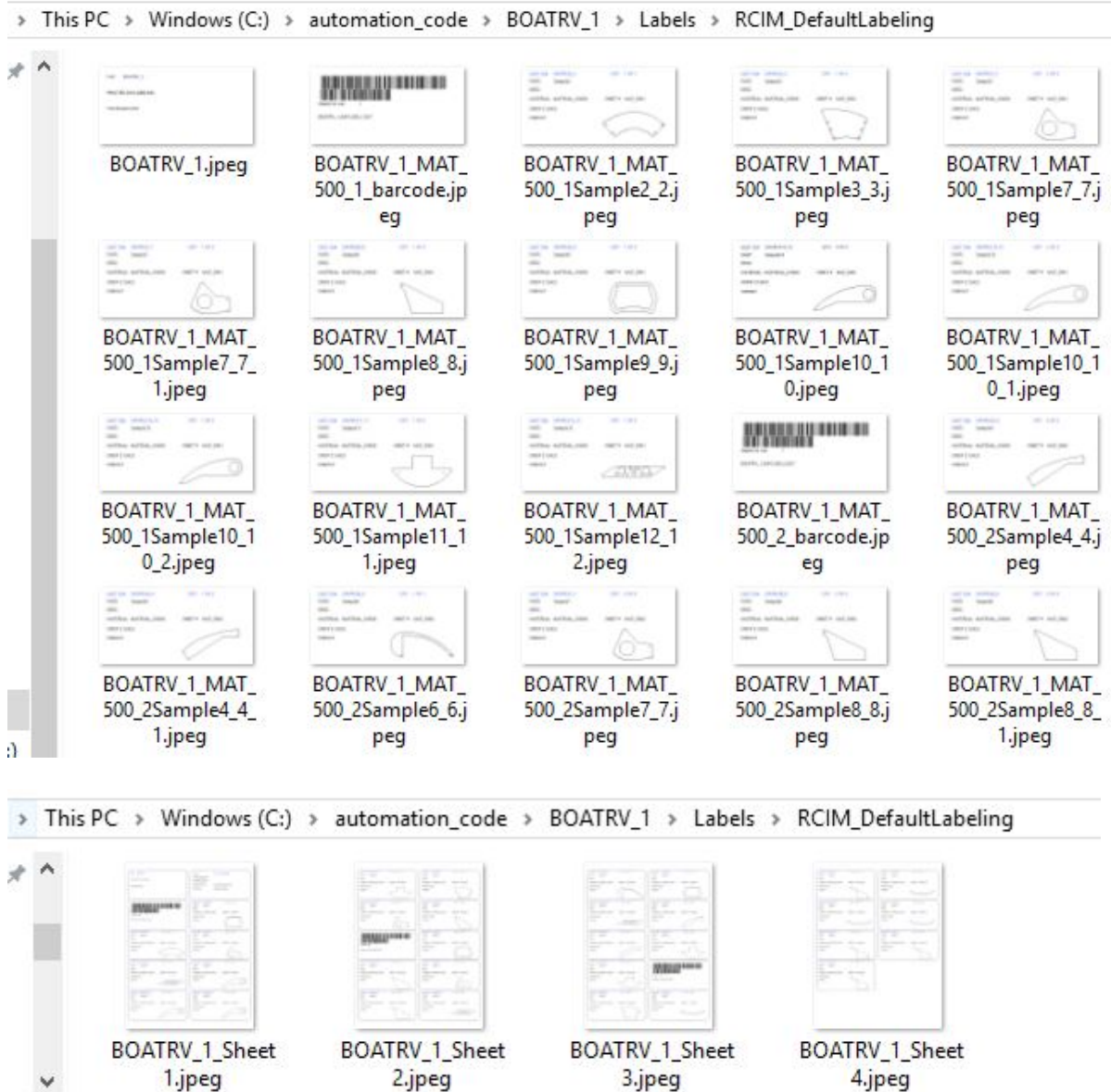
When 'Create Individual Labels' and/or 'Create Sheet Labels' is selected, you can output them as image files.

This screenshot shows the 'Generate Image File' checkbox in the 'Generate Image File' section highlighted with a red rectangle. The 'Image Format' is set to 'PDF' and the 'Naming Scheme' is 'PartName'. The 'Send to Printer' checkbox is still unchecked.

When generating image files, you will need to define:

- 1) DPI (dots per inch) for clarity
- 2) Type of Image Format
 - a) BMP
 - b) GIF
 - c) JPG
 - d) PDF
 - e) PNG
 - f) TIFF
 - g) WMF
- 3) Naming Scheme
 - a) PartName
 - b) Innoagg (only to be used by users that have this add-on option)

The 'Generate Image File' option will create the images in the Router-CIM Automation Suite output folder once the job has completed. From this folder, you can print or move the files.



Send to Printer

When 'Create Individual Labels' and/or 'Create Sheet Labels' is selected, you can output them directly to a printer.

By checking the 'Send to Printer' box, you will define the printer that you want the labels to be printed to and the DPI.

Options for DPI are:

- 1) High
- 2) Medium
- 3) Low
- 4) Draft

The screenshot shows the 'Settings Panel' with the 'Output Settings' tab selected. On the left, under 'Output Settings', are checkboxes for 'Create Individual', 'Create Sheet Labels', and 'Draw Border On Labels', all of which are checked. In the center, under 'Create Labels', are checkboxes for 'Create Job Label', 'Create Material Label', 'Create Part label', 'Create Scrap Label', and 'Create NC Code Label', all of which are checked. To the right of these are 'Generate Image File' (checked), 'DPI' (100), 'Image Format' (JPEG), and 'Naming Scheme' (PartName). On the far right, the 'Send to Printer' checkbox is checked and highlighted with a red box. Below it, the 'Printer' is set to 'Brother MFC-6490CW Printer' and the 'DPI' is set to 'Medium'.

The 'Send to Printer' option will print the labels from Router-CIM Automation Suite once the job has completed.

Automation Label Settings

Setting up a Router-CIM Automation Suite Job with a Label Configuration

Once the configurations have been created, you will then be able to select the label configuration you would like to use.

The screenshot shows the 'RCIM Labeling' dialog box. It contains a table titled 'Active Label Configurations' with three columns: 'Active', 'Name', and 'OutputTNP'. There are two rows of configurations. The first row has a checked box in the 'Active' column, the name 'RCIM_DefaultLabeling', and an unchecked box in the 'OutputTNP' column. The second row has an unchecked box in the 'Active' column, the name 'RCIMLabel', and an unchecked box in the 'OutputTNP' column.

Active	Name	OutputTNP
<input checked="" type="checkbox"/>	RCIM_DefaultLabeling	<input type="checkbox"/>
<input type="checkbox"/>	RCIMLabel	<input type="checkbox"/>

Under the 'Printing and Labels' tab in the job, you can select the configuration that you want to use for that job. Simply check the box in the 'Active' column to create the labels. More than one configuration can be selected if desired.

RCIM Labeling

Active Label Configurations		
Active	Name	OutputTNP
<input type="checkbox"/>	RCIM_DefaultLabeling	<input type="checkbox"/>
<input checked="" type="checkbox"/>	RCIMLabel	<input type="checkbox"/>

If you have the Touch-N-Print add-on by CIM-Tech, you can also create custom output labels for this system by selecting the check box for the configuration desired under the 'OutputTNP' column.

RCIM Labeling

Active Label Configurations		
Active	Name	OutputTNP
<input type="checkbox"/>	RCIM_DefaultLabeling	<input type="checkbox"/>
<input checked="" type="checkbox"/>	RCIMLabel	<input checked="" type="checkbox"/>

If you select an Active Label Configuration that you want to use for all jobs going forward, make sure to ['Save Current Settings as Job Defaults'](#) so that all future jobs will be created with the correct configuration selected.

Advanced Topics

Included in this section is commonly asked questions about certain processes and procedures within Router-CIM Automation Suite.

If you cannot find the answer to your question, please contact CIM-Tech Customer Support at support@cim-tech.com or 877-549-8211.

Pocketing in Automation

To develop pocketing cuts in Automation, you have to determine first if the pocket contains "islands" or not. An island is an area, inside of the pocket, that you do not want the router bit to cut. If the pocket has an island you want the cutter to avoid, then all the geometry of the pocket (outside shape of pocket and any internal islands) must be on a layer that starts with the letters "POCKET". Then associate that layer name to your knowledge using the pocketing cycle and you are set.

If your pocket does not contain islands, then the layer name can be any qualified layer name as long as it does not have the letters "POCKET" in it. Then Router-CIM Automation Suite will gather that layer, and cut each shape on that layer separately with the pocketing knowledge you associate to the layer.

If a part contains more than one (1) pocket with island feature, you need to combine the pocket and island features separately.

For instance, the first pocket with island would be associated to layer: Pocket1 and the second pocket with island feature would be associated with layer: Pocket2 and so on.

If your pocket does not contain islands, then the layer name can be any qualified layer name as long as it does not have the letters "POCKET" in it.

Alternating Tables on a Twin Table Machine

To alternate X/Y and X/V programs in Router-CIM Automation Suite with a twin-table post processor, you will need to add the following variable to the Router-CIM NCVAR file. To learn how to add variables to NCVARs, please refer to the [NCVar Editor](#) section.

Add the variable exactly as shown:

twinpost

Set the variable to T.

This assumes that the machine has two post processors, one for processing on the right table, and one for processing on the left table.

The NC code file produced for the left table will be different from that for the right table. Odd numbered programs will be for the left table, and even number programs will be for the right table.

This is also referred to as Pendulum Processing

Note: There must be _ALT.\$pp versions of the post installed.

Cutting files from a Spreadsheet in Router-CIM Automation Suite using Right-Click

To run Router-CIM Automation Suite jobs from an XLS or XLSX file, you must have a minimum of 3 columns, the **Part Name, Part Material, and Quantity**. Additional information can be in the XLS or XLSX. Some job/part settings can be set in the XLS or XLSX or as job defaults explained in step one below. Here is a list of items that can be imported from an XLS or XLSX:

Ignore the Column

Part name

Part material

Quantity

X dimension

Y dimension

Z dimension

Ignore panel

Backside (several values here affect how the part is cut: 0=nest part, 1=has an associated backside macro, 2=code as single part, 3=not used, 4=nest and code single part, 5=is an irregular stock shape)

Description

Label field 1

Label field 2

Label field 3

Label field 4

Label field 5

Label field 6

Label field 7

Label field 8

Rotate part

Rotate angle

Knowledge drawing

Doit file

Post processor

Print nests

Print single parts

Job name

Filler quantity

Start point on longest side

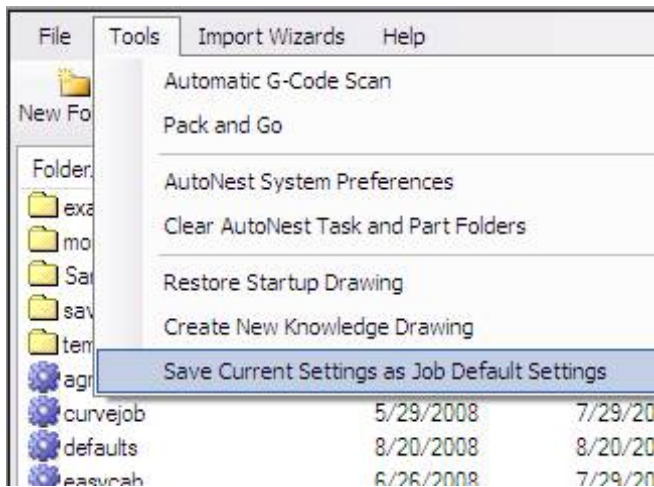
Nest Rotation

Variable (For Parametric Macros)

To use the right click, cut with Router-CIM Automation Suite on an XLS or XLSX file you must have some defaults set first.

1) Set up a Router-CIM Automation Suite job with the desired knowledge, DOIT, post processor, and nesting options.

Then pick Tools pull down, and Save Current Settings as Job Default Settings



2) Import an XLS or XLSX using the Import Wizard

Select the 'Excel and Comma Delimited' import

Set up each column to match the column information provided in the file.

Excel Import Wizard

Required Steps

Select file to import

Confirm worksheet: Sheet1\$

Number of header rows: 0

Assign column headings

Optional Features

Select a saved format

Save format (header rows and column headings)

Delete selected format

Refresh Data

	Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7
	Part Name	Part Material	Qty	X Dim	Y Dim	Z Dim	Job
	C:\rcim_work\sty...	3/4 MDF	12	16.6875	27.6875	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	6	18	19.25	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	7	18	5.5	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	5	14	8.25	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	3	14.375	8.25	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	7	18	20	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	8	21.375	18	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	3	18	22	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	2	21.5	26.25	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	6	21.25	32.25	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	8	16.375	24.25	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	12	14.375	25.5	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	3	21.375	30.25	0.75	exc...
	C:\rcim_work\sty...	3/4 MDF	4	16.375	15.5	0.75	exc...

Status: Idle Data File: C:\rcim_work\xls\DOOR_import1.XLS

3) Save this mapping layout and give it a name by picking 'Save format (header rows and column settings)'

Enter a name for this (your default) import format:

Saved Format Name

Please enter a name for this format.

OK

Cancel

door_importer

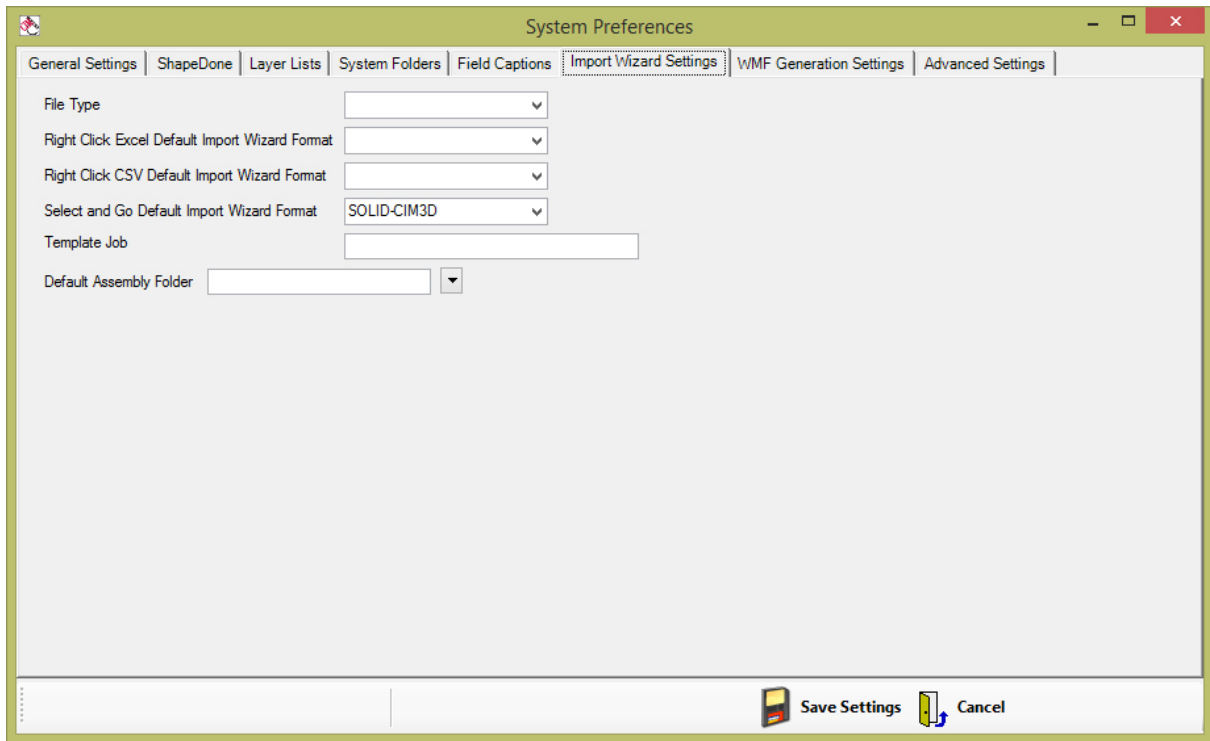
4) Set Router-CIM Automation Suite to use that default Import Wizard format.

Pick File, then Settings.

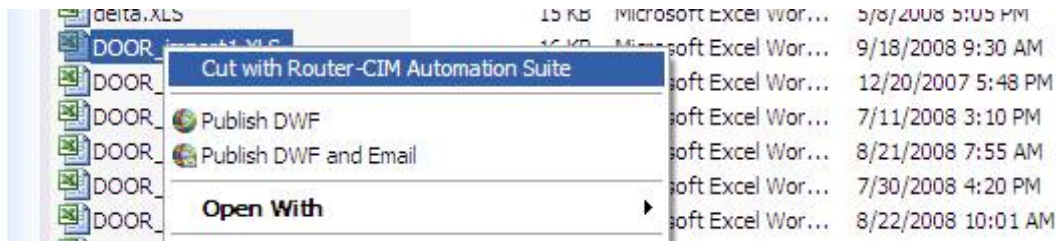
Then pick the '**Import Wizard Settings**' tab.

Select the correct Right-Click drop down for the type of file you are using and select the format that you saved in the previous step.

Exit Automation

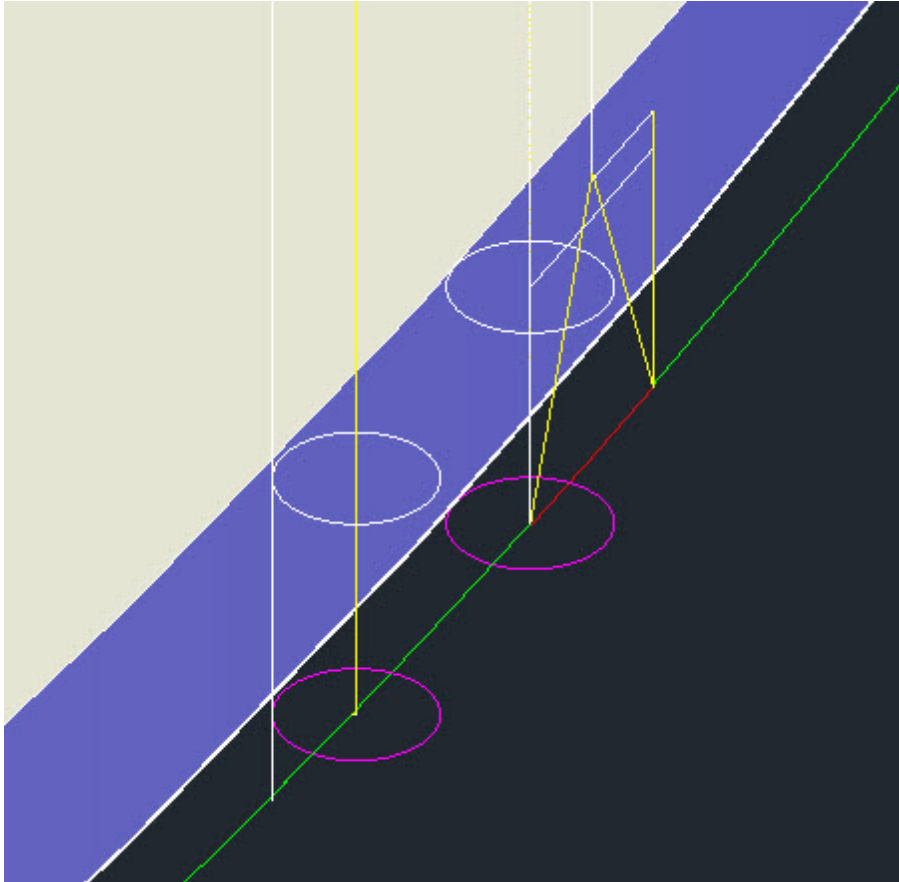
**5) Right click on an XLS or XLSX and pick 'Cut with Router-CIM Automation Suite'**

The format you set as default will be used to map out the XLS or XLSX file you selected. Router-CIM Automation Suite will build a job using the job defaults you set up in the first step, and then import the XLS or XLSX knowing what data is in each column because of the default import format that you set in step four.



Using the PREDRILL Function

The PREDRILL function is a feature that will create a drill hole at the lead-in of any plunge type cycle. This also includes the ability to use this feature on 2D tabbing during the lead-in. This will relieve the stress on the tool during a plunge movement allowing for a cleaner entry into a cut.



To utilize the PREDRILL function, you would need to create a knowledge using Drill Motions. The diameter of the drill that is selected will be the diameter of the circle that is created by the PREDRILL function. Use your 'Rank' to have the PREDRILL function happen prior to the knowledge that you are using it for. The circle drawn by the PREDRILL function will be placed on a layer in AutoCAD named NC_PREDRILL. The circle drawn by the PREDRILL function will not include thickness so the knowledge will need to have 'Total Cut Depth' derive the depth from the material or a preset depth. Save the knowledge so that it is available to Router-CIM Automation Suite.

The last step in order to utilize PREDRILL is to assign the knowledge that you created to the layer NC_PREDRILL in the DOIT file. The knowledge association you create needs to be the last association in the DOIT file. For more information on how to assign a knowledge to a defined layer, go to the ['Edit DOIT File'](#) section. You can use the 'Move Association' buttons in the DOIT file to make sure that the NC_PREDRILL association is located at the bottom of the list.



Move Association

<input type="checkbox"/>	27	TG_12516STAYDOWN	TG_12516STAYDOWN
<input type="checkbox"/>	28	TG060STAYDOWN	TG060STAYDOWN
<input type="checkbox"/>	29	TG9051STAYDOWN	TG9051STAYDOWN
<input type="checkbox"/>	30	TAB_DRILL	NC_PREDRILL

Kinematic Time Study

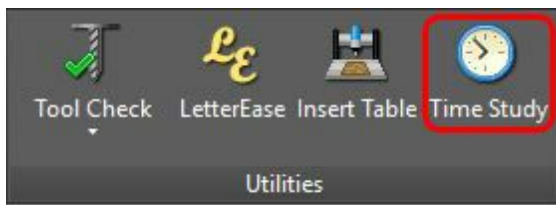
The Kinematic time and tool change calculations are used to report estimated cycle times for the sequence of tool paths run through Router-CIM. The calculations are based off the Kinematic equations, which can be used to determine the amount of time it would take to perform a motion.

Note: In order to use the Kinematic Time Study with Router-CIM Automation Suite, you may need to add the NCVAR (variable) to the system. For more information, click [here](#).

Note: In order to use the Kinematic Time Study with Router-CIM Interactive, you will need to select the option during the Sequencing process. For more information, click [here](#).

The time study parameters are accessible from inside of AutoCAD once Router-CIM has been loaded. On the Router-CIM ribbon from AutoCAD, the Time Study Parameters are located under the 'Utilities' panel.

Note: Router-CIM must be started within the AutoCAD drawing in order to access the Time Study Parameters. For more information on how to start Router-CIM within AutoCAD, click [here](#).



Select the 'Time Study' icon under utilities.

To set the parameters of the Kinematic Time Study, follow this guide.

The 'Time Study Parameters' window will be displayed:

Note: The Post Processor your selected will be the parameters that are shown. If you have multiple post processors, you will need to set up each one in order to get estimated cycle time results with different post processors.

Save: This option will save the current parameters for use with the selected post processor.

Exit: This option will close the Kinematic Time Study parameters. If changes had been made, you will be prompted to save the changes if needed.

2) Post Word Values

This section displays any post words that have been defined and the associated time value in seconds.

For more information on 'Post Word Values', click [here](#).

3) Machine Parameters

This section is where the index speed, machine acceleration rate, and the maximum machine angle can be set.

For more information on 'Machine Parameters', click [here](#).

4) Tool Change Times

These sections are where you set the amount of time it takes to perform a tool change to/from different types of tools.

For more information on 'Tool Change Times', click [here](#).

5) Maximum Machine Angle

This section is where you set the angle where your machine would need to come to a complete stop in order to accomplish.

For more information on 'Maximum Machine Angle', click [here](#).

Post Word Values

Post words are text based variables that are interpreted by the post processor, and can be used to generate NC code.

Finding Acceptable Post Words

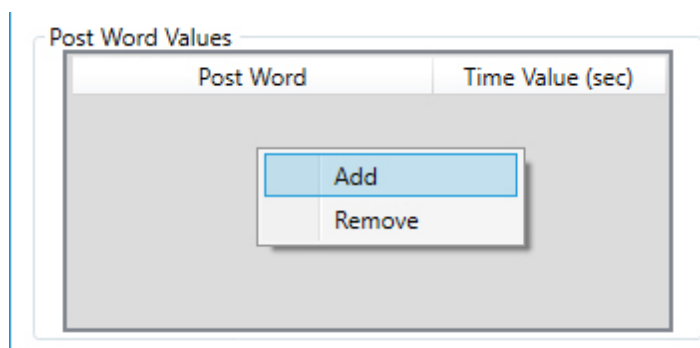
Post words are specific to each post processor, and in order to see the list of available post words, you can use the **POST** or **POSTCMDS** command once Router-CIM is loaded in AutoCAD. This will bring up a screen similar to the one shown below called "Inline Commands." The column labeled "Post Command" are acceptable words for that post processor, and could be used as a part of the Kinematic Time Study.

Note: You should also reference your post processor application notes for any additional post words.

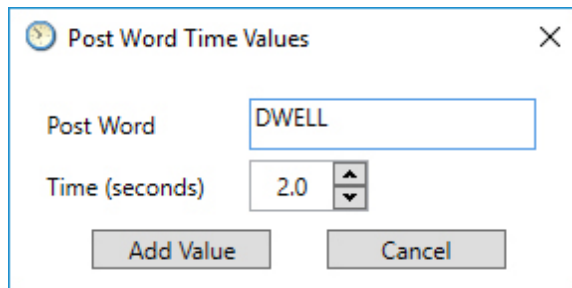
InLine Commands				
Post Command	Output Code	Format	Buffer	Modal
BBUF	T	5	None	Modal
BBUF2	T	5	None	Modal
BOREOFFSET	G52X	5.4	None	Not Modal
C	C	5.4	None	Modal
CIRCUL/CCLW	G3		None	Modal
CIRCUL/CLW	G2		None	Modal
CUTCOM/LEFT	G41		Before	Modal
CUTCOM/OFF	G40		Before	Modal
CUTCOM/RIGHT	G42		Before	Modal
CUTZ	Q	5.4	After	Not Modal
CYCDWL	P	5.2	After	Not Modal
CYCLE	G	2	Before	Not Modal
CYCZ	R	5.4	After	Not Modal
D	D	2	Before	Not Modal
DEPTH	Z	5.4	After	Not Modal
DEPTHX	X	5.4	After	Not Modal
DEPTHY	Y	5.4	After	Not Modal
DRLFED	F	5.2	None	Modal
DSPEED	S	8	None	Modal
DWELL	G4P	5	Current	Not Modal
DWELLX	G4X	5.4	Current	Not Modal
ENDP	M2		Current	Not Modal
FEDRAT	F	5.2	After	Modal
FINI	M99		Current	Not Modal

Adding a Post Word

Once you have found the post word that needs time associated to it in the summary report. Right-click in the "Post Word Values" box as shown below and select **'Add'**.



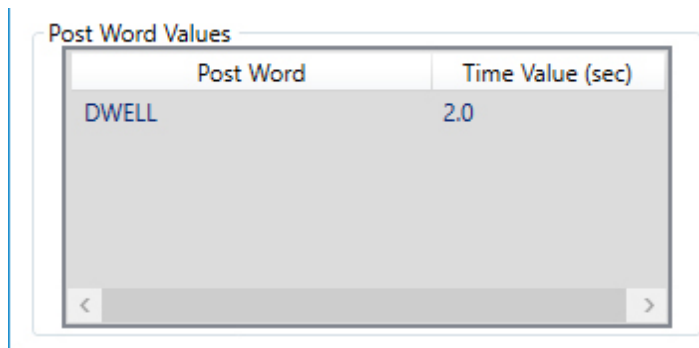
Once you have selected **'Add'**, the 'Post Word Time Values' window will be displayed:



A dialog box titled "Post Word Time Values" with a close button (X) in the top right corner. It contains two input fields: "Post Word" with the text "DWELL" and "Time (seconds)" with the value "2.0". Below these fields are two buttons: "Add Value" and "Cancel".

Enter the post word and associated time value you would like added. Next click the '**Add Value**' button.

Once you have added the value it will be displayed in the "Post Word Values" box. You can now add another post words at this time. When finished adding values click the '**Cancel**' button.



A window titled "Post Word Values" containing a table with two columns: "Post Word" and "Time Value (sec)". The table has one row with the values "DWELL" and "2.0".

Post Word	Time Value (sec)
DWELL	2.0

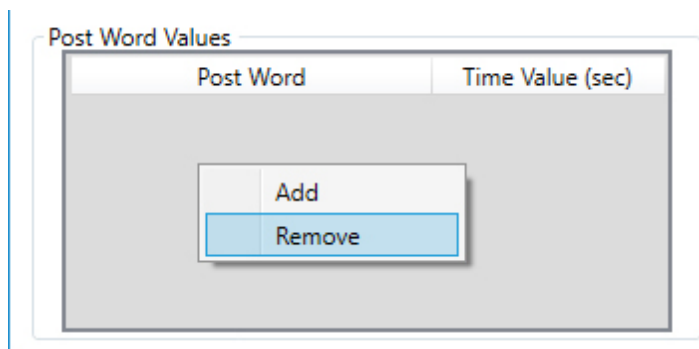
For Example:

Above is an example of adding the post word DWELL, which was found in the post application notes as an available post word, and associated a time of 2 seconds to it. During sequence every time the post reads the word "DWELL" in the cuts, the summary report will have 2 seconds added to it.

Note: This added time will be reflected under the index time for that specific tool in the summary report.

Removing a Post Word

In order to remove a post word, select the item you would like to remove. Once selected, right-click in the 'Post Word Values' box as shown below and select '**Remove**'.



A window titled "Post Word Values" containing a table with two columns: "Post Word" and "Time Value (sec)". The table is empty. A context menu is open over the table, showing two options: "Add" and "Remove".

Post Word	Time Value (sec)
-----------	------------------

Machine Parameters

This section is where the index speed and the machine acceleration rate are set.

Index Speed - Defines the speed at which the machine performs an index move (in/min or mm/min).

Machine Acceleration Rate - Defines how fast the machine accelerates (in/sec² or mm/sec²).

Machine Parameters

Index Speed

3150.0 in/min

Machine Acceleration Rate

148.8 in/sec²

Note: These values should be obtained from your CNC manufacturer.

Tool Change Times

These sections are where you set the amount of time it takes to perform a tool change to/from different types of tools. Please reference your Post Application Notes for what tool numbers are associated with each tool type. Each section should be reviewed and adjusted as needed in order to receive estimated cycle time studies.

Change From Tool to	Change From Drill to	Change From Piggyback to	Change From Aggregate to	Change From Other to
Drill: 10.0 sec	Drill: 1.0 sec	Drill: 10.0 sec	Drill: 10.0 sec	Drill: 10.0 sec
Tool: 10.0 sec	Tool: 3.0 sec	Tool: 10.0 sec	Tool: 10.0 sec	Tool: 10.0 sec
Piggyback: 10.0 sec	Piggyback: 3.0 sec	Piggyback: 10.0 sec	Piggyback: 10.0 sec	Piggyback: 10.0 sec
Aggregate: 10.0 sec	Aggregate: 10.0 sec	Aggregate: 10.0 sec	Aggregate: 10.0 sec	Aggregate: 10.0 sec
Other: 10.0 sec	Other: 3.0 sec	Other: 10.0 sec	Other: 10.0 sec	Other: 10.0 sec
Tool #'s 1 To 199	Drill #'s 301 To 399	Piggyback #'s 201 To 299	Aggregate #'s 1000 To 9999	

Note: The first tool of the sequence will use the tool change time value found under "Other" for that section of Tools.

For Example:

In the below section 'Change From Tool to' we are defining the amount of time it takes to go from a "Tool" in the main spindle to one of the following types of tools Drill, Tool in the main spindle, Piggyback, Aggregate, Other. If the first tool used in a sequence is a "Tool" then the tool change time will be 15 seconds. If the next tool used is another tool in the tool changer, then the tool change time will be 7 seconds.

Change From Tool to

Drill	20.0	sec
Tool	7.0	sec
Piggyback	20.0	sec
Aggregate	20.0	sec
Other	15.0	sec

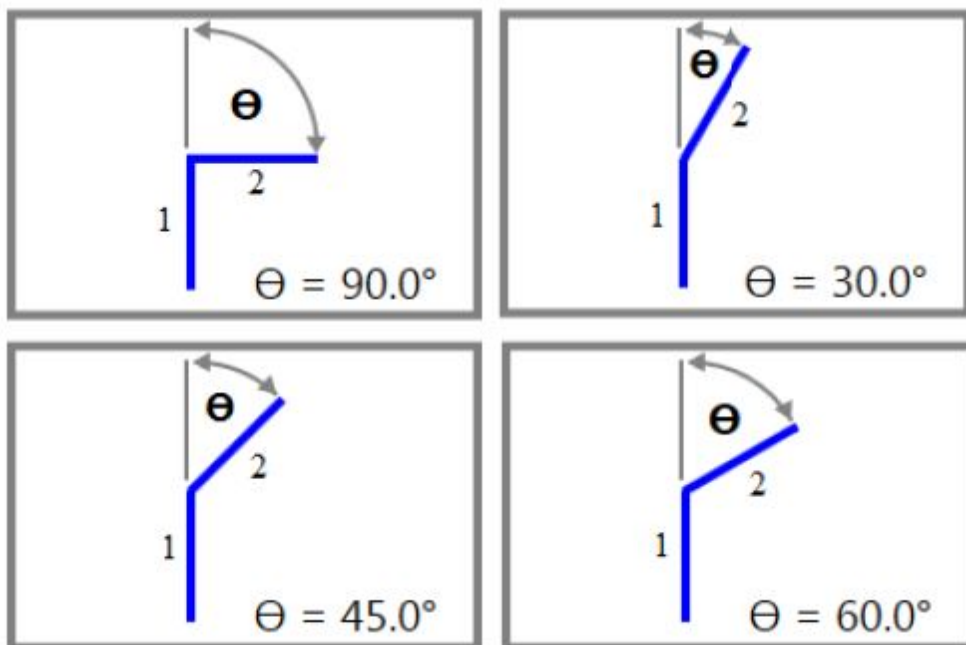
Tool #'s To

Maximum Machine Angle

As the maximum machine angle changes the image to the right will adjust. Below are 4 examples showing the 90, 60, 45, 30 degree angles.

What is being represented in these images is that if the starting line segment is "1" and the next line segment is "2" that this is the angle that is being changed between those 2 line segments.

The maximum angle that can be inputted into this field is 90 degrees, because it is assumed that if the machine needed to make a change in direction that is greater than or equal to 90 degrees that it would need to decelerate to a feed rate of 0.



For Example:

In this scenario we are cutting an outside shape at a feed rate of 350 in/min. Some sections of the outside shape causes the machine to have a change in direction of 45 degrees, 50 degrees, 70 degrees, and 90 degrees.

The maximum machine angle for this machine is 50 degrees.

The 45 degree and 50 degree change in direction will have no deceleration, and the feed rate of 350 in/min will be maintained.

The 70 degree change in direction is greater than what our machine can do without needing to slow down. There will be some deceleration performed to the feed rate to accommodate this change in direction. The decelerated feed rate is calculated based on the cosine of the change in direction angle.

The 90 degree change in direction is assumed to cause our machine to decelerate to 0 in/min in order to accomplish this turn.

About NCVARs

There are occasions when the "standard" installation of Router-CIM may not be applicable to your machine tool. Usually this can be fixed using the NCVARs, which are specific variables that Router-CIM uses to control the program.

You should only change the NCVARs when instructed to either by this manual or after consulting CIM-Tech. An improper setting could cause a failure in Router-CIM that may be hard to track down.

NCVARs Appendix

There are some standard variables that ship with the program that may see regular use in customizing the system to your needs.

The NCVAR will be first and in bold print. It's definition will be below it and in regular print.

Common Variables in Router-CIM Automation Suite

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Control how Router-CIM Handles an Ellipse

If the AutoCAD geometry is an ELLIPSE when you list it, then you must set ***NOPELLIPSE*** to **T** in Router-CIM in the NCVARS.

If the AutoCAD geometry is a POLYLINE when you list it, then you should set ***NOPELLIPSE*** to **nil** in Router-CIM in the NCVARS.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Changing the File Extensions in Router-CIM Interactive

Modify the two variables:

Under System Variables, select the variable ***ncfilext***, change the value to the appropriate file extension. Note the file extension has to be between two quotation marks.

i.e. "txt", "out", "NC", etc.

Under System Variables, select the variable ***prev_ext***, change the value to the appropriate file extension. Note the file extension has to be between two quotation marks.

i.e. "txt", "out", "NC", etc.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Manually place part labels using layer Nestcenter

If you want to define the location of the part label that Router-CIM Automation Suite uses for the placement of the part label on the part in the nested drawing, you can use this procedure.

Add the variable ***nestpoint*** and set the value to **T**. For instructions on how to define the location using a point in AutoCAD, go to the AutoCAD Help file.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Adjusting Pin Trim Variables for Two-Sided Nesting

To adjust the variables that affect the two-sided nesting PIN_TRIM function, you will need to add the following variable to the Router-CIM NCVAR file.

Add the variable exactly as shown:

pin_offset

pin_offset - overrides the distance of the cut from the edge. The top-side NC code file will be offset by this amount. Default is ½ the edge allowance of the nested material.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Adjusting Spacing Variable for Two-Sided Nesting

To adjust the spacing between the front side nest and the backside nest during the Two-Side Nesting function, you can add the variable ***2sidedspacing***. The default that Router-CIM Automation Suite uses is the materials bottom edge allowance.

Add the variable exactly as shown:

2sidedspacing

2sidedspacing - Changes the spacing between the front side nest and the backside nest during the Two-Side Nesting function. The acceptable value is a real number such as 0.5. This will leave a 0.5 gap between the front side nest and the backside nest. To disable this feature, set the variable to a value of nil.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Adjusting Pin_Trim Origin for Two-Sided Nesting

To adjust the pin_trim origin without affecting the sheet origin.

Add the variable exactly as shown:

sheet_origin_override

sheet_origin_override - "1" = Lower Left Corner
 "2" = Upper Left Corner
 "3" = Upper Right Corner
 "4" = Lower Right Corner

Note: The value entered **MUST** be between quotation marks.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Adding a Dwell/Pause at the end of a Drill Cycle

To add a dwell/pause at the bottom of a Drill Motions cycle, you will need to adjust the following setting in the Drill Motion cycle:

Change Position 17 in the Drill Motions cycle to DWELL/1 (or whatever time you want to pause, in seconds). You will get a G04P1 in the code after the tool is at cut depth.

Alternately on newer posts, you can use DWELLX/.1 and get G04X.1.

To know which option is right for you, please review your post's Application Notes.

To make this change, go to the Router-CIM Control panel and select the button for '**Mod Cycle**'.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Rectangular Part Checking

To have Router-CIM check if a part is a rectangle to allow Router-CIM to set the rotation of the part to 0 90 for more efficient nesting, you will need to add the following variable to the Router-CIM NCVAR file.

REC_CHECK

Set to T - This will allow the rotation on the part to be set to 0 90 if the part is a rectangle

Set to nil - This will not change the parts rotation and the rotation will be defined by the material the part is nested on

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Using the Kinematic Time Study with Automation

In order to use the Kinematic Time & Tool Calculation option through Router-CIM Automation, you may need to set an NC variable (NCVARS).

kinematic

Set to T - This will allow the Router-CIM Automation to use the Kinematic Time Study.

Set to nil - This will not allow the Router-CIM Automation to use the Kinematic Time Study.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Using the NOEXPAND variable for Veneer Match

The variable *NOEXPAND* controls the Veneer Match spacing between the matches by following the Veneer Match Location Points exactly as defined. You will need to add the following variable to the Router-CIM NCVAR file.

NOEXPAND

Set to T - This will allow the Router-CIM Automation to follow the Veneer Match Location Points exactly.

Set to nil - This will allow the Router-CIM Automation to follow the material's bridge width or Veneer Match Bridge as defined.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

Using the Cut After Nest feature with Router-CIM Automation Suite

The variable *cut_after_nest* uses the 'Backside' section of the DOIT file. This feature will utilize the 'Backside' DOIT file after the nest has been completed to allow for cuts to be applied after nesting has laid out the parts on the sheets of material..

Note: Using the *cut_after_nest* variable will disable some features such as Automated Two-Side Nesting and may add additional processing times during the job runs.

cut_after_nest

Set to T - This will allow Router-CIM Automation Suite to use the 'Backside' DOIT file after the nest has been completed.

Set to nil - Router-CIM Automation Suite will follow the standard DOIT file setup.

This variable should be added to the **System** section.

For information on how to add variables to the NCVAR file, refer to the [NCVAR Editor](#) section.

GEOMETRIC VARIABLES

brng

Largest gap in data for BLEND if NCJOIN is "T" to close

cstrang

Start angle (expressed in degrees) for circular shapes. 0 starts at 3 o'clock.

cutfil

Offset fillet radius for rounded corner filleting in CUT.

endmrng

End margin on slicer for 3D surface machining

leadfed

This is the percentage of the programmed feedrate that the lead in will be performed at.

leadratio

A value used in an equation to determine the angle of entry for Heli-lead cycles.

ncjoin

Use polyline join with *brng* tolerances to close gaps in geometry commonly caused by DXF or IGES transfers.

offvect

NOT USER CONFIGURABLE

pat_fuzz

Value for pattern recognition tolerance amount

r4x_corner_incr

This variable controls the Router-CIM 4th axis interpolation cycle. This value controls how long the segments should be when interpolating around an inside corner.

r4x_corner_offset

This variable controls the Router-CIM 4th axis interpolation cycle. This is the distance from the inside corner the inside corner cuts will start.

reduceopt

To reduce surface machining tool path to arcs the response is "r" and "s" for no reduction to arcs.

reducezero

Data reduction tolerance of slicer for producing arcs in 3 axis machining.

LAYER VARIABLES

Bndclr

Boundary color

Cutclr

Cut line color for tool path other than CW and CCW.

Defclr

Geometry color for NC_GEO.

Inxclr

Index line color.

Inxfr

Index from layer.

Inxfclr

Index from color.

Inxlt

Index line type.

Inxto

Index to layer.

Inxtocl

Index to color.

Meshclr

Color for offset meshes.

Ncbackplot

Back plot layer.

Ncboundary

Boundary layer.

SYSTEM VARIABLES

3dtab

Turns Tabs into Helical ramps

3dhelix

If True, a polyline is created on layer NC_Helix and turned into a 3D arc in code.

If nil, a polyline is created on layer NC_leads and turned into point to point lines in code.

4aratiohp

Should always be T. Affects how 4 axis cutting treats complex geometry

4xramp

NOT USER CONFIGURABLE

5axis

"T" response will produce augmented data in the toolpath for full 5 axis solutions. This should only be set to "T" when a full 5 axis solution is provided by CIM-TECH.COM, Inc..

jobcomment

Always set to True. This provides the ability for Router-CIM to output comments in the code based on drawing parameters like drawing name, login name, date, etc.

no_fdtm

Eliminates the feed distance to material move which is the extra Z move before a cut starts. FEED DIST TO MATERIAL in the tool edit

sub_text

Text Height for Subroutine Labels.

adspost

If True, use the ADS version of the postprocessor.

auto_sub_num

Turns on automatic sub program numbering. When True, sub programs are automatically numbered, starting at 100 and increments by 100 for each additional sub.

cycmode

Canned cycle cutting mode, either 98 or 99. When set to 98, tools retract to Safety plane, When set to 99, tool retracts to R point set in Router-CIM control panel.

cycname

If True, all cut blocks have a name based on the drawing name. If nil, all cut blocks have a name based on the cycle name.

cyctype

Canned cycle type, on most Fanuc controls, use either 81 (spot drill cycle) or 83 (peck drill cycle).

delgeos

If set to T, the geometry objects used to make shapes are deleted when the shapes are made. If set to NIL (the default), the geometry objects are left on the NC_GEO layer when shapes are made.

doorgrain

Door-CIM VAR

doorplot

Door-CIM VAR

editlist

Name of the text editor for List Edit. For R10 and R11 users only.

heli_rotate

If set to a non zero value, the lead in and lead out of heli cycles will be rotated away from the part by the specified amount. The leads will only rotate when a single pass cut is made.

linesort

Only use lines for open shape group sorting.

ncfilext

Extension to use on the output file.

Valid input is a string (value in quotes) up to 3 characters.

ncfillloc

The path to use when creating the NC Code file. If blank, uses the current drive and directory.

ncfont

Text font for NC text.

ncoffset

NIL = uses AutoCAD offset, T = uses Router-CIM offset

ncorigin

If True, Router-CIM will require a placement of an origin symbol and axis description during a CUT cycle. Mainly used for Nesting applications.

noshpchk

If set to T, no checks are made for duplicate definitions of selected objects. If set to NIL (the default), checks are made during the define process to see if the selected objects have already been defined. If so, messages instruct you exit or redefine the objects.

nsee

If T, enables the prompts for the Tape-to-Part extension. If you do not have NSEE or WNSEE, leave this nil.

offbias

This number is added to the offset value in Router-CIM. Sometimes this bias is required to created a specific offset.

OTHER VARIABLES

TS_NO_NBL

NOT USER CONFIGURABLE

dwfmake

This variable will create a dwf image of a part created by the Parametric Macro Builder. Setting it to T will cause the drawing to zoom to extents and a print window will flash. Automation can then preview this part. Set to nil if Automation is not going to be used.

emul_mode

NOT USER CONFIGURABLE

emul_tool_percent

NOT USER CONFIGURABLE

geo_color

NOT USER CONFIGURABLE

nc_lines

NOT USER CONFIGURABLE

nestvec

Scale factor for nest shape definition. Set to 1 for normal operation.

AutoNest Manual

For information on how to set up basic '[Nesting](#)' and '[Advanced Nesting](#)' in Router-CIM Automation Suite, refer to the appropriate section in the manual.

AutoNEST is addressed to the common problem faced in a number of industries where repetitive or non-repetitive parts have to be cut economically from given stock sheets. This optimization requirement is often encountered in industries such as shipbuilding, aerospace, sheet metal, garment manufacture, furniture work, etc.

While each different industry may have its own characteristics, the underlying objective of the manufacturer is to maximize utilization and minimize material wastage through judicious layout of parts.

A part's geometry is input in a simple *TEXT* .*VEC* file format or in the AutoCAD *DXF* format.

Through a Task editor (***TaskEdit*** command), you can define your nesting requirements, i.e.: stock sizes, quantity of each stock size, parts to be nested, quantity and grain orientation of each part, cutting gap and so on. Taking into consideration these parameters, *AutoNEST* will search through the numerous possible arrangements and present near-optimum layout solutions.

AutoNEST assists the fabricator in maximizing material utilization and reduce drastically the time taken to plan a layout.

The program has been highly optimized to provide the shortest run time, a fraction of the time of what it would have taken manually.

AutoNEST features a single intelligent nesting engine - **NestPRO** to handle all your nesting jobs.

AutoNEST allows the incorporation of an "intuition" factor from the experienced designer so as to take into consideration the practical features only known by the designer. Nested layouts can readily be edited through AutoCAD or any CAD system for the best final layouts.

From Version 9.5.1 onwards, *AutoNEST* has incorporated an additional functionality called **Auto-Bridge** into the software.

© COPYRIGHT 1989 to 2018 RADAN SYSTEMS PTE LTD All Rights Reserved

This publication or any parts thereof, may not be reproduced by any method, for any purpose, without prior written consent of RADAN SYSTEMS PTE LTD. For conditions and permission to reproduce this material in other languages other than English language, please contact RADAN SYSTEMS PTE LTD.

Warranty, Liability and Disclaimer

RADAN SYSTEMS PTE LTD makes no warranty, either expressed or implied; including, but not limited to any implied warranties of merchantability or fitness for a particular purpose, regarding these materials. In no event shall Radan Systems Pte. Ltd be liable to anyone for special, collateral, incidental or consequential damages in connection with or arising out of purchase or use of these materials. The sole and exclusive liability to RADAN SYSTEMS PTE LTD, regardless of the form of action, shall not exceed the purchase price of the materials described herein.

RADAN SYSTEMS PTE LTD reserves the right to revise and improve its products as it sees fit. This manual describes the state of this product at the time of publication, and may not reflect the product at all times in the future.

Trademarks

AutoNEST is a registered trademark of Radan Systems Pte Ltd.

NESTPRO is a registered trademark of Radan Systems Pte Ltd.

AutoCAD is a registered trademark of Autodesk Inc.

Windows XP /Windows Vista /Windows 7 /Windows 8/ Windows 10; Windows 2000 Server / Windows 2003 Server /Windows 2008 Server and Windows 2012 Server are the registered trademarks of Microsoft Inc.

In no circumstances may any of the trademark or copyright notices or proprietary legends contained on or within the software or documentation or on their packages be removed.

Getting Started with AutoNest

You can run AutoNEST within Windows or AutoCAD. The minimum AutoCAD version is 2010. Please check with your authorized AutoNEST Reseller regarding the latest AutoCAD version to run with your copy of AutoNEST.

This section of the documentation makes reference to the use of AutoNEST in conjunction with AutoCAD. AutoCAD is used both as a graphical input and output for AutoNEST.

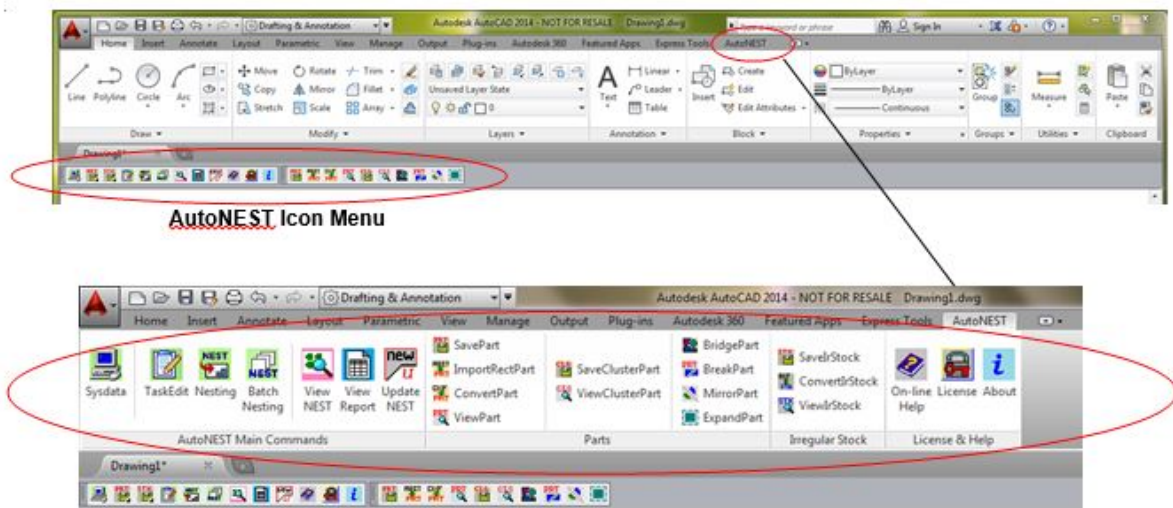
AutoCAD Environment

The step-by-step work flow of *AutoNEST* in the AutoCAD environment is as follows:

(1) Starting AutoNEST

To run **AutoNEST**, click the **Start** button (located at the lower-left corner of screen) of your Windows and select **Programs** → **AutoNEST for AutoCAD 200X**.

This will launch AutoCAD and AutoNEST. Next, you will see an additional pulldown menu titled “**AutoNEST**”, located between the “Modify” and “Window” pulldowns.



(2) **Create Parts**

There are a number of commands under Parts that will enable you to either save or convert your files /data into AutoNEST Parts. They are :-

- ✍ **SavePart, MirrorPart** – save your AutoCAD drawing (must not be a block but AutoCAD entities) into AutoNEST Parts.
- ✍ **ConvertPart** - converts DXF files (parts in individual DXF files) into AutoNEST Parts.
- ✍ **ImportRectPart** - imports rectangular parts defined in Microsoft Excel or Text files into AutoNEST Parts.
- ✍

(3) **Create Task**

This is the task definition stage. Through Windows dialog boxes, the particulars of parts required to be nested onto specified stock sheets are specified. Details such as edge allowances, cutting gaps and other nesting criteria are being specified here. The command name is **TaskEdit**.

(4) **Nesting**

After having defined the nesting requirements in the previous stage, **Nesting** can now be invoked to compute the layout solutions. The resulting layouts are automatically converted into **DXF** format and displayed onto the current AutoCAD session. Two ASCII text files, two XML files and one Excel file are created to reflect the nesting results.

(5) **UpdateNEST**

Once a nested layout is displayed onto the current drawing, you can further edit it based on your better judgment and experience using AutoCAD's move, copy, rotate or delete commands. This process will be invisible to *AutoNEST*, as such *AutoNEST* needs to be "updated" so that new utilization percentages of the stock sheet can be displayed. This is achieved via **UpdateNEST** command.

(6) **ViewNEST**

This command allows you to review layouts of previously nested tasks within current Task/ Parts directory.

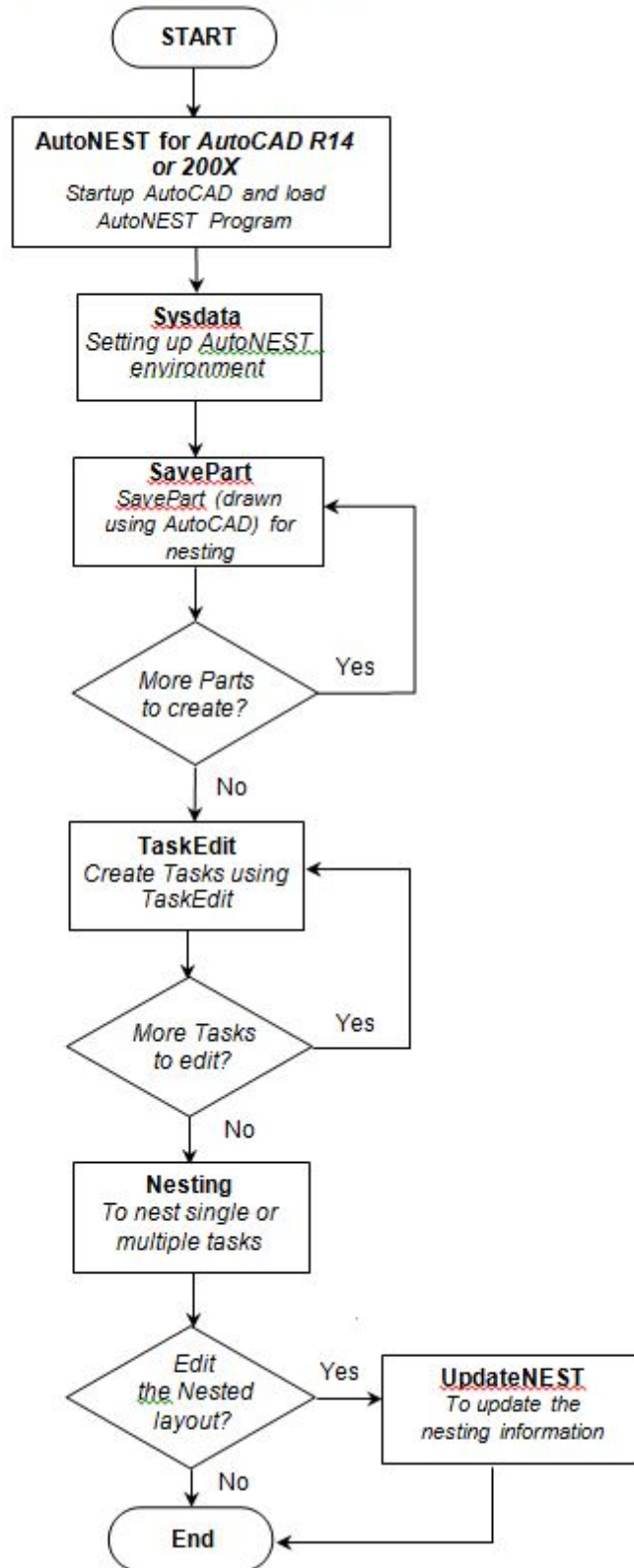
(7) **Auto-Bridge**

You can define the Bridge specifications in **TaskEdit**. To view the bridged nested layouts, select either **ViewNEST** or **Nesting / BatchNesting** commands.

See [Command Flow Chart](#) of *AutoNest*










Command Flow



Flow Chart of AutoNEST






Command Overview

Below is a list of *AutoNEST* icon commands available within the AutoCAD. The same commands are available from the AutoNEST pulldown as well.


Sysdata - Environment Settings	
	Sysdata To set environmental settings for running <i>AutoNEST</i> .
Part	
	SavePart To save a Part which you have constructed or drawn using AutoCAD <i>DRAW</i> commands.
	ImportRectPart To import user data (.TXT, .XLS...etc) of rectangles into AutoNEST's Parts (.VEC) and Task (.JOB) .
	ConvertPart To convert DXF files to Parts (.VEC). As well as from Part (.VEC) files to DXF format.
	ViewPart To view a saved Part, click this icon. At the AutoCAD command prompt you will be able to view a part in either .DWG /.DXF /.VEC formats.
	SaveClusterPart To save two or more <i>AutoNEST</i> parts into a cluster.
	ViewClusterPart To view a Cluster-part within AutoCAD.
	BridgePart To bridge any selected parts from an AutoNEST nested layout.
	BreakPart To break any selected part into two or more smaller parts (.VEC).

	<p>MirrorPart</p> <p>Besides saving a part which you have constructed or drawn using AutoCAD DRAW commands, MirrorPart command will also save a mirror of the part with a user-definable name.</p>
	<p>ExpandPart</p> <p>To create a part with an expanded profile on the AutoCAD screen. If the part is expanded with the cutting gap, this command can help you to manually edit the layouts by fitting the part onto some available space.</p>




Irregular Stock



	<p>SaveIrStock</p> <p>To save an irregular stock which you have constructed or drawn using AutoCAD DRAW commands</p>
	<p>ConvertIrStock</p> <p>To convert DXF files to irregular stocks (.stk). As well as from irregular stock (.stk) files to DXF format.</p>
	<p>ViewIrStock</p> <p>To view a saved irregular stock, click this icon.</p>

Task Edit


	<p>TaskEdit</p> <p>To specify or edit a “task” by specifying a list of parts and stocks to be nested and the associated parameters of the nesting requirements.</p>
---	--

Nesting and ViewNEST



	<p>Nesting</p> <p>To nest one particular task and to display the nested layout.</p>
	<p>BatchNesting</p> <p>To nest multiple tasks in one go.</p>
	<p>ViewNEST</p> <p>To view the nesting layouts of previously nested tasks.</p>

	ViewReport To view nesting summary report (in Excel format) of the nested tasks.
	UpdateNEST To update the nested layouts after editing

On-line Help

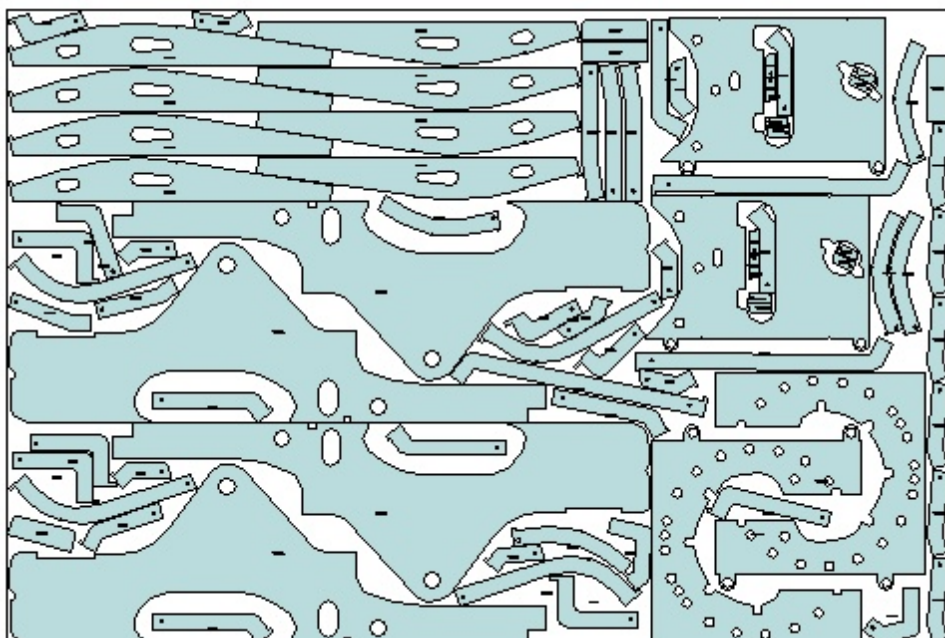
	On-line Help To view on-line help file, click this icon.
---	--

License & About

	License To display the current license status.
	About AutoNEST To view the version number of the program, click this icon.

Sample Nests

Samples of typical nested layout:





Commands

The Commands are broken up into 10 sections:

- 1) [Sysdata](#)
- 2) [Parts](#)
- 3) [Irregular Stock](#)
- 4) [Task Edit](#)
- 5) [Nesting](#)
- 6) [View Nest](#)
- 7) [Update Nest](#)
- 8) [AutoNest On-Line Help](#)
- 9) [AutoNest License](#)
- 10) [About AutoNest](#)

AutoNest Sysdata

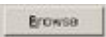


Sysdata is an important command to set the working environment for running AutoNEST. Before you begin to work with any of the other AutoNEST commands, it is recommended that the **Sysdata** command be activated to set the working environment.

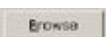
Select the **Sysdata** command and the following dialog box will be displayed.

Directory Setting

Parts/ IR-Stock Directory

To type in or click the  button to choose a directory where your parts and irregular stocks are to be located. (The directory or sub-directory must be created first)

Task Directory

To type in or click the  button to choose a directory where your tasks are to be located. (The directory or sub-directory must be created first)

Input Part /IR-stock – Layer Setting

Outer Profile Inner Profile

These are the layer, color and the parts' leadin/leadout "FILTER" settings that you can specify for the following commands.

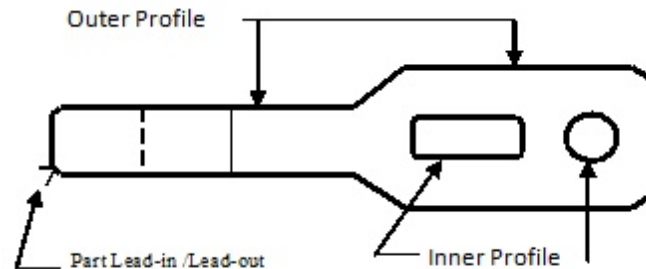
- **SavePart & SaveIrStock**

Part Leadin

- **ConvertPart & ConvertlrStock**

These settings are especially helpful if the parts that you are saving or converting contain marking lines that touch or intersect the external /internal profiles of the shapes.

By specifying the layer/ color/ part leadin filters, you will help the above commands to differentiate profiles that are crucial to nesting (outer and inner profiles of the part) against those that are not (markings or folding lines).



You can specify more than ONE colors in the “Color” field by entering for example “1,3” for colors 1 and 3 (separated by a comma)

Nested Layout Presentation

Units

The units setting of inputs and outputs. There are 4 choices to choose from:

- Metric
- Architectural Imperial (1' 3-1/4")
- Decimal Imperial (15.25")
- Engineering Imperial (1' 3.25")

Accuracy

The number of decimal places or the number of digits to the right of the decimal point (0 to 4).

If the units chosen above is Imperial, the denominator of the fractions or the accuracy will be expressed as follows:

- 1 for full integers, no fractions
- 2 for 1/2" (half)
- 4 for 1/4" (quarter)
- 8 for 1/8" (eighth)
- 16 for 1/16" (sixteenth)

File Format

This option decides which part file format is to be used when generating graphical nested layout on screen. Three choices are available :

DWG
VEC
DXF

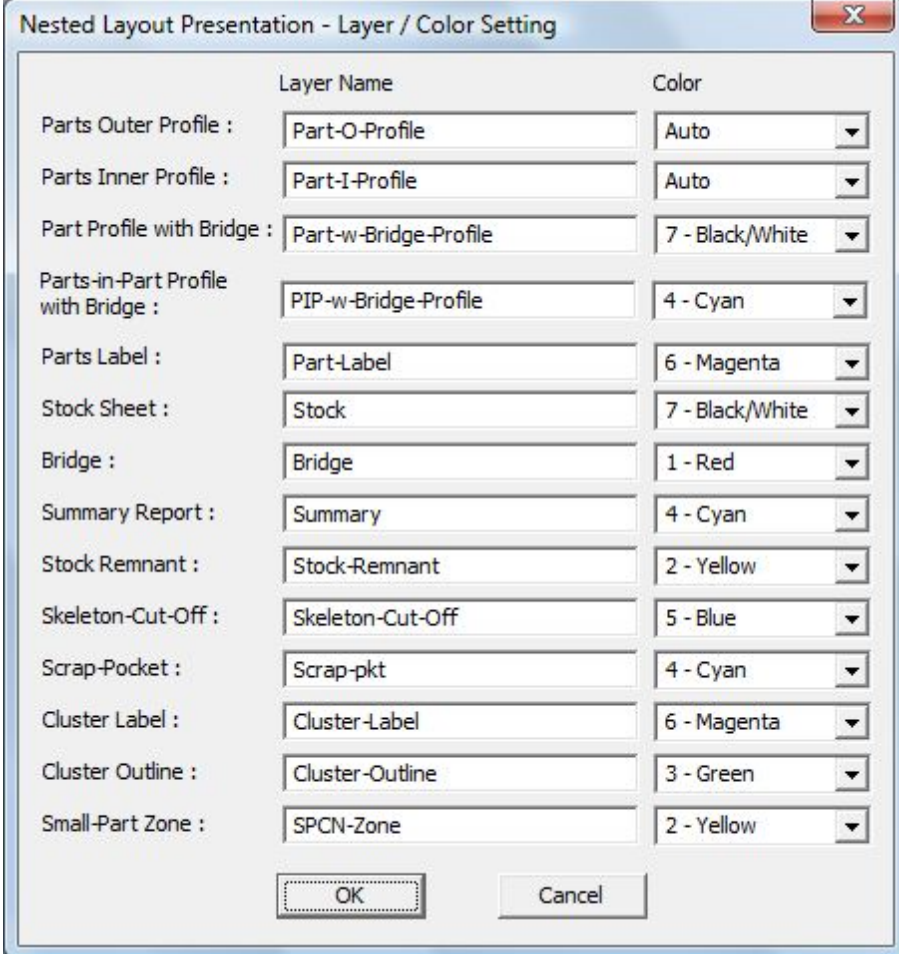
If you have chosen VEC format, please note the “Nested Layout – Layer Settings” (click the “Layer Setting” button) to define the layers and colors of the nested layout.

If you have chosen DXF or DWG format, the nested layouts will display the layers/ colors of your parts as they were originally created.

Layer / Color

The following dialog box will appear when you click this

button  button



The dialog box titled "Nested Layout Presentation - Layer / Color Setting" contains a table with two columns: "Layer Name" and "Color". It lists various layout elements and their corresponding layer and color settings. At the bottom, there are "OK" and "Cancel" buttons.

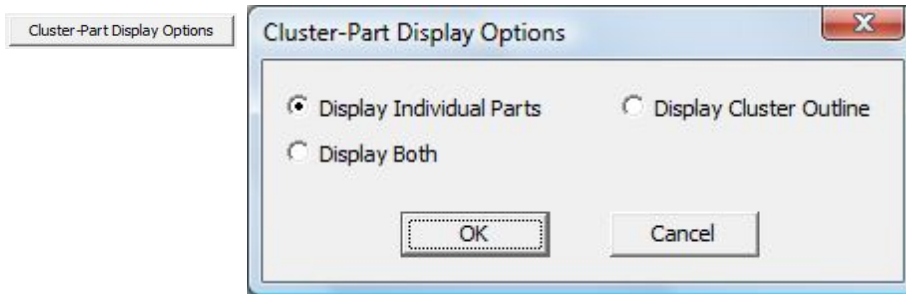
	Layer Name	Color
Parts Outer Profile :	Part-O-Profile	Auto
Parts Inner Profile :	Part-I-Profile	Auto
Part Profile with Bridge :	Part-w-Bridge-Profile	7 - Black/White
Parts-in-Part Profile with Bridge :	PIP-w-Bridge-Profile	4 - Cyan
Parts Label :	Part-Label	6 - Magenta
Stock Sheet :	Stock	7 - Black/White
Bridge :	Bridge	1 - Red
Summary Report :	Summary	4 - Cyan
Stock Remnant :	Stock-Remnant	2 - Yellow
Skeleton-Cut-Off :	Skeleton-Cut-Off	5 - Blue
Scrap-Pocket :	Scrap-pkt	4 - Cyan
Cluster Label :	Cluster-Label	6 - Magenta
Cluster Outline :	Cluster-Outline	3 - Green
Small-Part Zone :	SPCN-Zone	2 - Yellow

User can define the layer and color for Part Labels, Stock Sheet and Summary reports.

PARTS

If the **Sysdata** "File Format" is VEC, the Parts displayed on the nested layout will follow the layer and color settings here. If DWG or DXF "File Format" are chosen, the Parts' layer and color settings will be exactly the same as they were originally saved.

See diagram below for explanation of each of the names.

**Display Individual Parts**

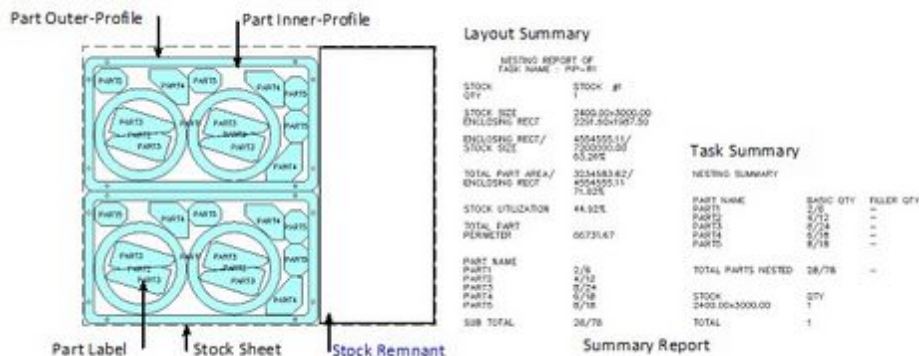
If this option is chosen, the individual Parts of each Cluster-part will be displayed. This is the default option.

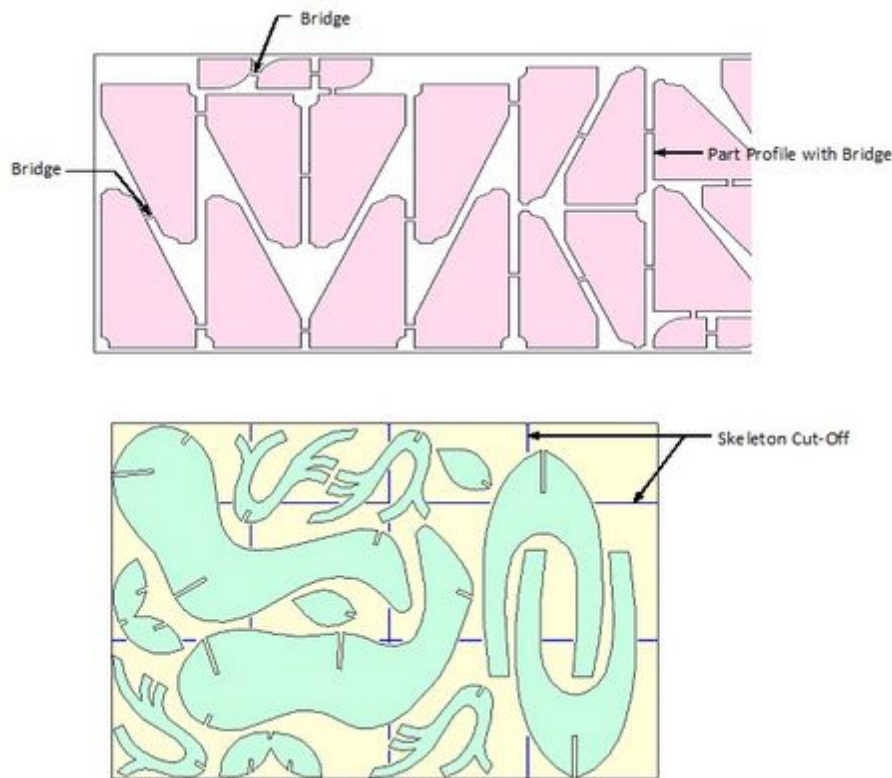
Display Both

If this option is chosen, the individual Parts and the outline of each Cluster-part will be displayed.

Display Cluster Outline

If this option is chosen, only the outline of the Cluster-part is displayed.





Layout Summary Task Summary

Layout Summary – report on the nested results of a particular nested layout. By default this report is displayed at the lower right corner of each nested layout.

Task Summary – report summary on the nested results of a Task (.job). By default this report is displayed next to the Layout Summary of the last nested layout.

Mark the relevant check-boxes if you wish to display the reports and set the Text Size of the Reports when displayed on screen.

See the Illustrations on top of Page 5-4.

You can change the default position of these reports in ANEST.SET @LAYOUT.

To change the layer/color of the Reports, click the “**Layer/ Color Setting**” button

Text Size

Text Size of the Reports as displayed on screen.

Label Parts

Mark this check-box if you wish to display Part Labels and set the Text size of the Part label when displayed on screen.

To change the layer/ color of the Part labels, click the “**Layer/ Color Setting**” button.

Text Size

Text size of the part label when displayed on screen.

“Auto” means the text size displayed will always be proportional to the size of the part.

Label Repeated Parts

Mark this check-box if you wish to add a part Label on each and every part on the nested layout.

When this check-box is un-marked, when there are say 10 parts of the same name nested, only ONE of the 10 is being labeled.

Layout Summary

Layout Summary – report on the nested results of a particular nested layout. By default this report is displayed at the lower right corner of each nested layout.

Task Summary

Task Summary – report summary on the nested results of a Task (.job). By default this report is displayed next to the Layout Summary of the last nested layout.

Mark the relevant check-boxes if you wish to display the reports and set the Text Size of the Reports when displayed on screen.

You can change the default position of these reports in ANEST.SET @LAYOUT.

To change the layer/color of the Reports, click the “**Layer/ Color Setting**” button

Text Size

Text Size of the Reports as displayed on screen.

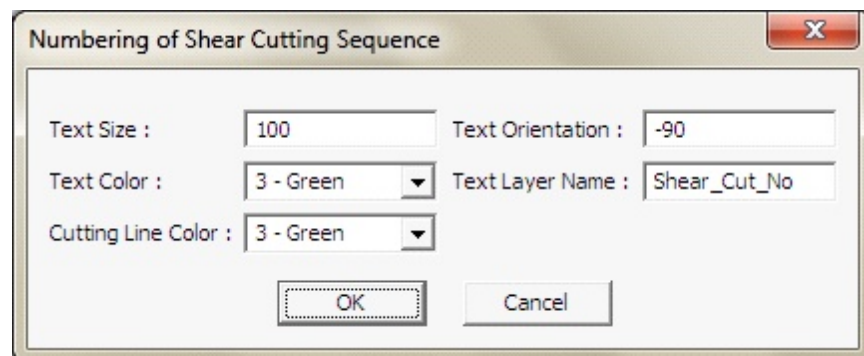
Display Repeated Layout

Mark this check-box if you wish to display the same nested layout repeatedly.

Shear Cut No.

You are able to pre-define the text size, layer and color of the Auto-Numbering of the cut sequence/Cutting lines of shear-cuts here.

When this “Shear Cut No.” button is clicked, the following dialog box is displayed.



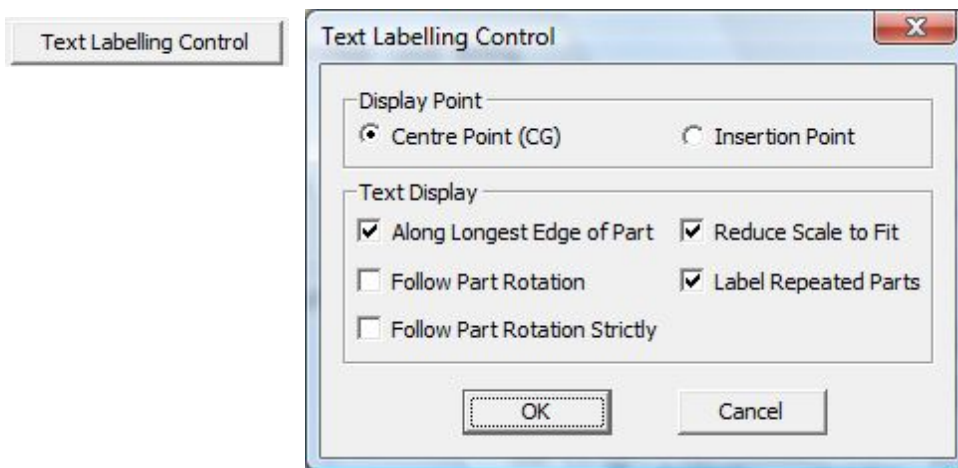
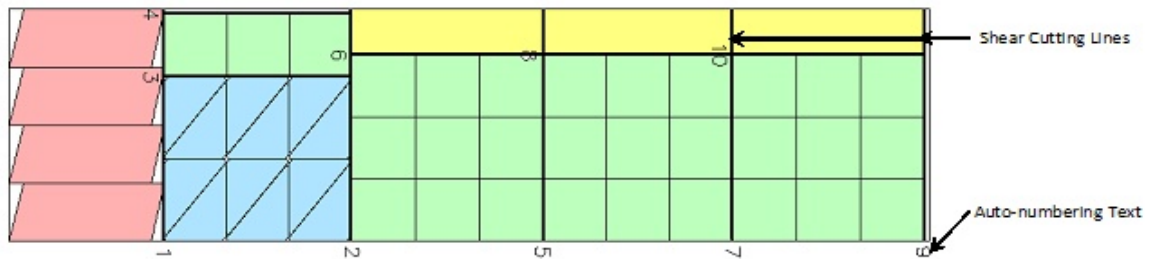
The dialog box titled "Numbering of Shear Cutting Sequence" contains the following fields and controls:

- Text Size :** A text input field containing the value "100".
- Text Orientation :** A text input field containing the value "-90".
- Text Color :** A dropdown menu showing "3 - Green".
- Text Layer Name :** A text input field containing the value "Shear_Cut_No".
- Cutting Line Color :** A dropdown menu showing "3 - Green".
- Buttons:** "OK" and "Cancel" buttons at the bottom.

You can preset the text size, color, orientation and layer name of the Auto-numbering of shear cutting sequence, as well as the color of the Cutting lines.

(By default, the Cutting lines will assume the same layer as the Text).

See illustration below :



Display Point

The location or display point of the text label. Two options are available :-

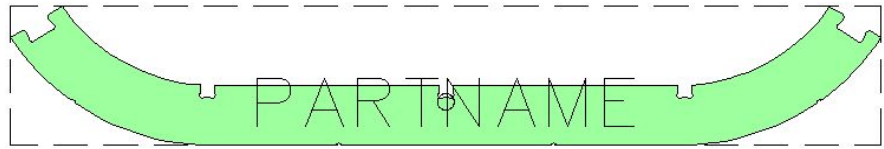
- Centre Point (Centre of Gravity) or
- Insertion Point.

By default the text will be displayed "middle" justified from the display point.

Along Longest Edge of Part

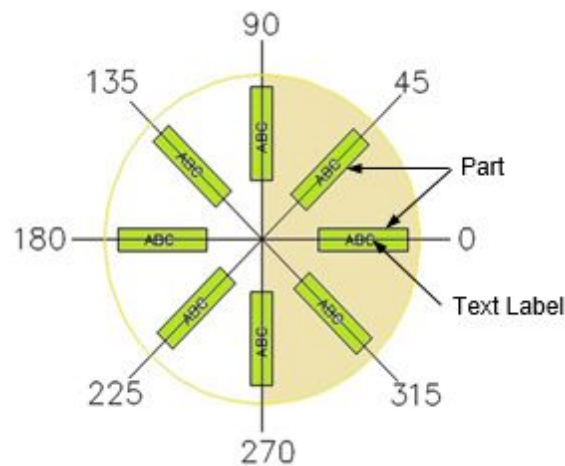
"Along Longest Edge of Part" means that the text label will be displayed along the longer side of the Part's smallest enclosing rectangle (from the "Display Point"). See examples below :-



**Follow Part Rotation**

Text will be displayed following the rotation angle of the Part. However, there is a display convention adopted for this setting. For examples :-

- 0 and 180 deg. are displayed as 0 deg
- 45 and 225 deg. are displayed as 45 deg
- 90 and 270 deg. are displayed as 90 deg
- 135 and 315 deg. are displayed as 315 deg..... and so forth.



If BOTH "Along Longest Edge of Part" and "Follow Part Rotation" / "Follow Part Rotation Strictly" check-boxes are marked, by default "Along Longest Edge of Part" setting will take precedence.

Follow Part Rotation Strictly

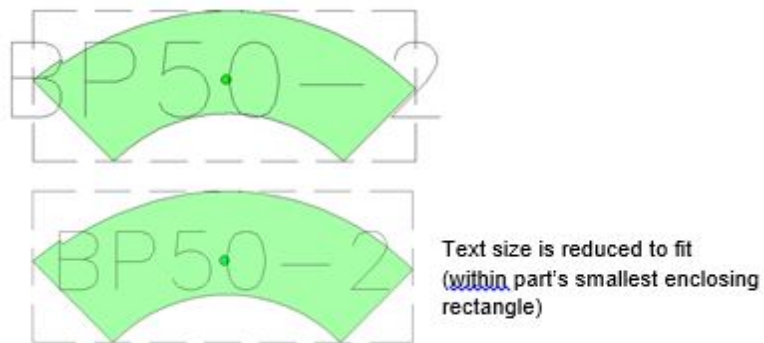
Unlike the previous "Follow Part Rotation", this Text display option will display text following the rotation angle of the Part strictly.

Therefore text will be displayed 0 degree if the part rotation is 0 degree.

Similarly text will be displayed 180 degrees (upside down) if the part is rotated 180 deg.

Reduce Scale to Fit

If the part label is outside of the part's smallest enclosing rectangle and if this setting is ON, then AutoNEST will automatically scale-down the text size such that the label is displayed within the part's smallest enclosing rectangle. See examples below :-



AutoNest Parts

This section introduces eight commands related to **PART** within the AutoCAD environment. They are used to save, expand or view parts that have been drawn or edited using AutoCAD commands. In addition, .DXF files can be converted into Parts as well.



[SavePart](#)

To save a Part which you have constructed or drawn using AutoCAD *DRAW* commands.



[ImportRectPart](#)

To import user data (.TXT, .XLS...etc) of rectangles into AutoNEST's Parts (.VEC) and Task (.JOB) .



[ConvertPart](#)

To convert DXF files to Parts (.VEC). As well as from Part (.VEC) files to DXF format.



[ViewPart](#)

To view a saved Part, click this icon. At the AutoCAD command prompt you will be able to view a part in either .DWG /.DXF /.VEC formats.



[SaveClusterPart](#)

To save two or more AutoNEST parts into a cluster.



[ViewClusterPart](#)

To view a Cluster-part within AutoCAD.



[BridgePart](#)

To bridge any selected parts from an AutoNEST nested layout.

**BreakPart**

To break any selected part into two or more smaller parts (.VEC).

**MirrorPart**

Besides saving a part which you have constructed or drawn using AutoCAD DRAW commands, MirrorPart command will also save a mirror of the part with a user-definable name.

**ExpandPart**

To create a part with an expanded profile on the AutoCAD screen. If the part is expanded with the cutting gap, this command can help you to manually edit the layouts by fitting the part onto some available space.

Save Part



Parts are generally components that have been constructed or drawn using AutoCAD **DRAW** commands. Once parts are designed and drawn with AutoCAD, **SavePart** command can be used to save the parts under the part directory as specified in **Sysdata**.

Within the AutoCAD drawing session where the parts to be nested had already been drawn, select the **SavePart** command either from the *AutoNEST* pulldown or icon menu. The following prompts will appear sequentially at the command prompt:

Save Part :
Part Name <> :
Require to save holes in Part? (Y/N) <Y> :
Insertion Pt :
Select objects :
Proceed to save another Part? (Y/N) <Y> :

Part Name <> :

This is to prompt for the name of the new part. (max. no. of characters - 31, SPACE is not accepted)
File already exists. OK to overwrite? (Y/N): <N>

This prompt will be displayed if the entered part name already exists.

Require to save holes in Part? (Y/N) <Y>:

This will enable the **SavePart** command to save internal profiles of a part as holes, which may be used to nest other parts.

Insertion Pt:

This is the pick-up point of the part as well as the location for label or tag of the part. Insertion point is recommended to be one of the vertices of the part or within the geometry of the part. AutoCAD object-snap modes under the **OSNAP** command, such as **Endp** or **Midp** are allowed.

Select Objects:

The Entity Select modes of AutoCAD are applicable here. You can type **W** (Window), **C** (Crossing), **R** (Remove) or simply pick the required entities to be saved as part.

If **W** (Window) is entered at this stage, a dynamic window will be formed to enclose the relevant geometry of the part. All the entities within the window will be saved as an *AutoNEST* part.

Proceed to save another Part? (Y/N) <Y> :

This prompt will allow you to save another part without quitting the command.

SavePart Essentials:

- a) **. VEC** , **. DWG**, and **. DXF** files are created, where :
 - **VEC** is a text file which record geometric information for a part, such as x, y coordinates of vertices, bulge values for arcs, center point, coordinates and radius for circles.
 - **DWG** is the drawing file generated by AutoCAD when a part is saved as a **Block/WBlock**. This is to keep a true copy of the part geometry submitted to *AutoNEST*.
 - **DXF** is a Drawing Interchange File format to assist in interchanging drawings between AutoCAD and other programs.
- b) **Text and Dimensions** are filtered out in the **. VEC** file but are retained in the **.DWG** and **.DXF** files for final presentation.
- c) **Markings** : If the Part has markings, make sure you set the 'Layer Name' & 'Color' filters of the parts so that **SavePart** is able to differentiate the part profiles from the markings. You define the layer names and colors in **Sysdata** "Input Part /IR-Stock – Layer Setting".

Conditions for Part Profile

Acceptable	Unacceptable
<ul style="list-style-type: none"> Lines, Arcs, Circle, Rectangle, Polyline, Polygon and Spline entities 	<ul style="list-style-type: none"> Pedit-Spline and Pedit-Fit (acceptable if "Explode")
<ul style="list-style-type: none"> External profile and internal holes profiles of a part must be closed. 	<ul style="list-style-type: none"> Additional line/plines along the profile
<ul style="list-style-type: none"> Block entity (acceptable if "Explode") 	<ul style="list-style-type: none"> Crossing over on each profile or between profiles
	<ul style="list-style-type: none"> Part with more than 1500 vertices per profile (Inclusive of starting and ending vertices of arcs)

Import Rectangle Part



Important :

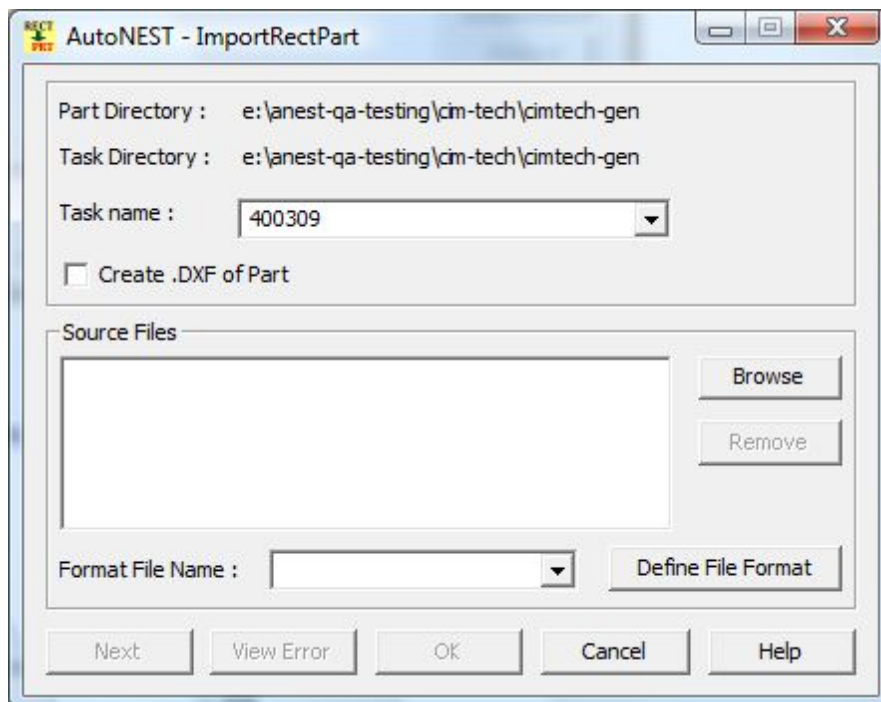
Please note that this particular command requires a copy Microsoft Office/ Excel 2000 or higher installed in your computer – in order to run. Otherwise an error message will be displayed.

ImportRectPart is a function that allows the user to import their data of rectangles in text or MS Excel format files (.TXT, .XLS or any text format files) into AutoNEST's Parts (.VEC files) and Task (.JOB).

If the user requires the rectangles to be displayed in the nested layouts as DXF files instead of VEC files, use the [ConvertPart](#) command to convert the .VEC files to .DXF files.


Select the **ImportRectPart** command icon from the icon menu or from the AutoNEST pulldown menu.

The following dialog box will appear:



Each dialog box input option is described in the following:

- Parts Directory** Display the default part directory as set in the **Sysdata**. In this case, it is also the target directory where all imported rectangular parts will be saved.
- Task Directory** Display the default task directory as set in the **Sysdata**. By default the task created by the **ImportRectPart** will be saved in this directory.
- Task Name** You can enter up to max. 31 characters (space and dot characters are not accepted) for the task name.

Enter a new task name or select an existing task. For more information about Task, see Chapter 4.5 TaskEdit.
- Create .DXF of Part** Mark this checkbox if you wish to create a .DXF file for each of the imported rectangular part. Otherwise leave it un-marked.
- Source Files** Click the  button to point to the directory that contains your data file(s).

You can select more than one files (but make sure that the files are of the same format)

To un-select the files, highlight the file(s) on the list box and click the "Remove" button.
- Format File Name** You can select from the drop-down list if an Import Format filename has been saved previously.
- Define File Format** This button will enable you to define the Import file format for your data file(s). A pop-up window will be displayed.

Format File Name Displays the selected Format filename if one has been selected.

Source File Definition Select from the drop-down list one of the three file types:

- Delimiter
- Excel
- Fixed Field

File Format Type

Delimiter - A Delimited file is where each field is separated by a special character. For example, comma-delimited (.csv) file where “comma” is the character that separates the different fields.

See the following example:

```
P7B-19,600,1200,7
P7B-20,900,950,1
P7B-22,1100,1400,2
P7B-34,1470,885,1
P7B-36,1360,1200,1
P7P2-1,750,900,10
P7X-1,1620,850,1
```

Excel - Microsoft Excel spreadsheet

Fixed Field - Text format where the data is arranged in well-defined “columns”. Example :-

```
P7B-19 600 1200 7
```


P7B-20	900	950	1
P7B-22	1100	1400	2
P7B-34	1470	885	1
P7B-36	1360	1200	1
P7P2-1	750	900	10
P7X-1	1620	850	1

Start Row The first starting row number of the rectangular parts. Only numeric inputs accepted.

Delimiter Character Only applicable if you have chosen the "Delimiter" format. For example, if the delimiter is a comma, enter ",", without the quotes.

Remarks Character The special character(s) that denote "remarks". Obviously rows with these remarks will be ignored during *ImportRectPart*.

Units You can choose either "Metric" or "Imperial".

End Row The last or end row number of the rectangular parts. Only numeric inputs accepted.

Field Name The essential field names required by the command. Some are mandatory, while others are optional.

- Part Name (mandatory)
- X-Dimension "
- Y-Dimension "
- Quantity "
- Filler Quantity (optional)
- Orientation "
- Priority "

Field Column For each field name, indicate the corresponding Column Number as in the data source file(s).

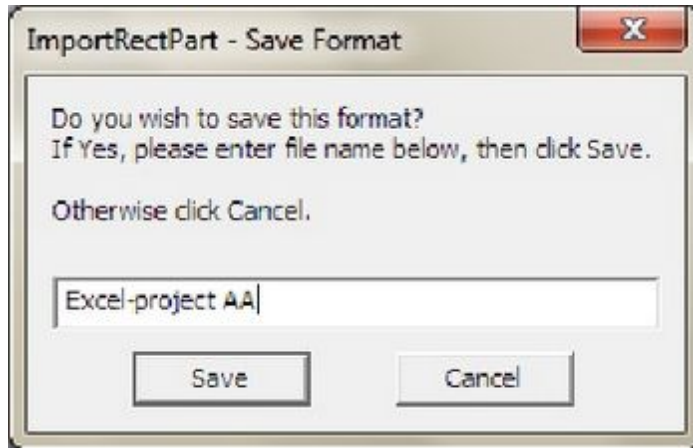
Where the 'File Format Type' is "Excel", you can enter the column number / alphabet.

Where the 'File Format Type' is "Fixed Field" or "Delimiter", you should enter the column / field number. Please see the following "Fixed Field" example :-

Col.1	Col.2	Col.3	Col.4
P7B-19	600	1200	7
P7B-20	900	950	1
P7B-22	1100	1400	2
P7B-34	1470	885	1
P7B-36	1360	1200	1
P7P2-1	750	900	10
P7X-1	1620	850	1

A rectangular button with a thin border and the word "Clear" in the center.

This button will clear all inputs in the dialog box, enabling you to re-enter from scratch.



You will see the above “**Save Format**” pop-up window to enable you to save the File Format definitions into a file for future use.

In the event that you encountered **Error Messages** during the *ImportRectPart* process, take note of the following :-

1. Rows numbers indicated in the error messages
2. The specific data on the erroneous rows.
3. The corresponding “Field Name” and “Field Column” specifications in the “Define File Format”.
4. Double-check the above-mentioned inter-related information and make the necessary changes.
5. Try again.

Convert Part

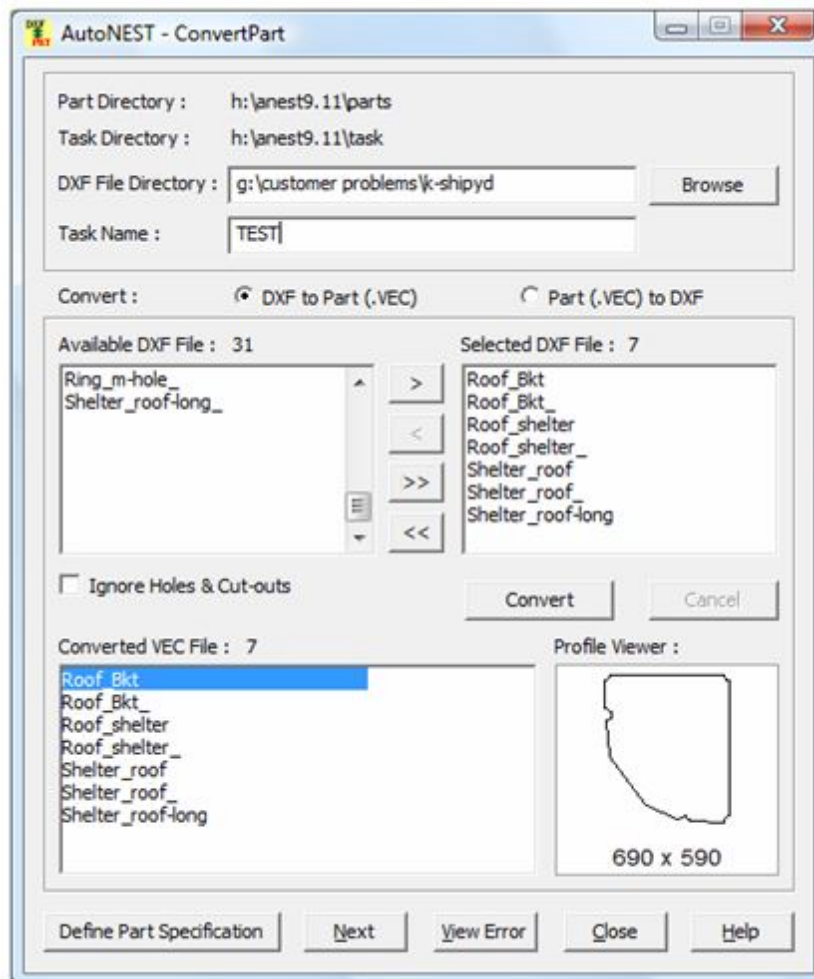


ConvertPart is a function that allows the user to convert part profile in DXF file format to Part (.VEC file format).

Similarly, it allows user to convert Parts (.VEC file format) back into DXF format.

If the Parts have markings, make sure you set the ‘Layer Name’ & ‘Color’ filters of the parts so that **ConvertPart** is able to differentiate the part profiles from the markings. You define the layer names and colors in **Sysdata** “Input Part /IR-Stock – Layer Setting”.

Select the **ConvertPart** command icon from the icon menu or from the AutoNEST pulldown menu. The following dialog box will appear:



Each dialog box input option is described in the following listing:

Parts Directory

Display the default part directory as set in the **Sysdata**. In this case, it is also the target directory where all converted file will be saved.

Task Directory

Display the default task directory as set in the **Sysdata**. By default the Task created by **ConvertPart** will be saved onto this directory.

DXF File Directory

To type in or click on the  button to choose a directory where the source file are located.

Task Name

You are able to define the Taskname in this dialog. This name will be carried forward to **TaskEdit**.

Convert DXF to Part (.VEC)



Check this radio button to convert DXF file format to Part (.VEC).

Convert Part (.VEC) to DXF




Check this radio button to convert Part (.VEC) file format to DXF.

Available DXF/VEC File

Display a list of available files found in the Part Directory (This will depends on which conversion has been chosen).

You can select one or more files to be converted by highlighting the filenames and then the  button or  button to select all files.

Selected DXF/VEC File

Display a list of files selected for the conversion. You can select the filenames and click the  button to remove the files from the "Selected Parts" list (but the files still remained on the "Available Parts" list) Or click the  button to remove all files. Click the  button to initiate the process of conversion.

Ignore Holes and Cut-outs

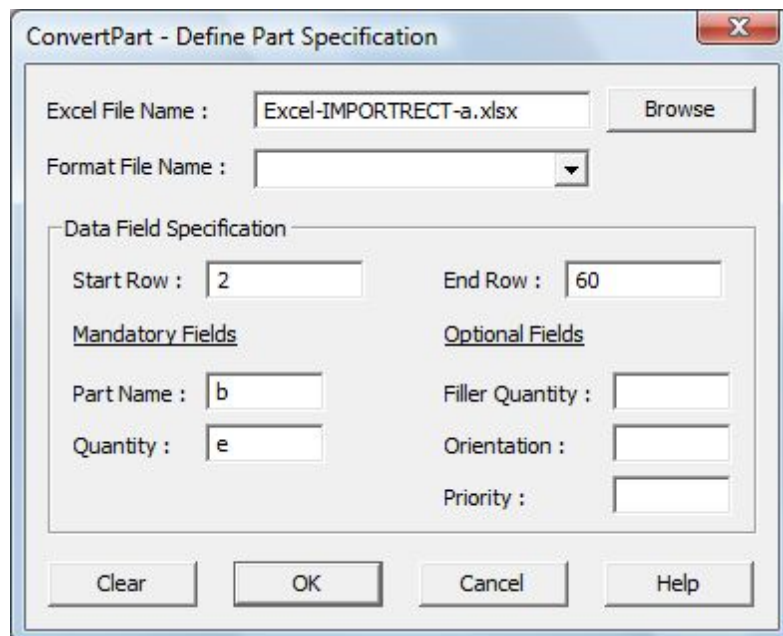
If this check-box is marked, holes and cut-outs will not be converted.

Converted VEC/DXF File

This will show files that have been converted successfully.

Define Part Specification

This button will enable you to define the part quantity, orientation constraints, part priority ...etc for each and every converted parts via an Excel spreadsheet. The following dialog will appear when this button is clicked.



The dialog box titled "ConvertPart - Define Part Specification" contains the following fields and controls:

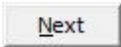
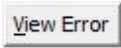
- Excel File Name :** A text box containing "Excel-IMPORTRECT-a.xlsx" and a **Browse** button.
- Format File Name :** A dropdown menu.
- Data Field Specification:** A section containing:
 - Start Row :** Text box with "2".
 - End Row :** Text box with "60".
 - Mandatory Fields:**
 - Part Name :** Text box with "b".
 - Quantity :** Text box with "e".
 - Optional Fields:**
 - Filler Quantity :** Text box.
 - Orientation :** Text box.
 - Priority :** Text box.
- Buttons:** **Clear**, **OK**, **Cancel**, and **Help**.

Excel File Name

Click the 'Browse' button to select the Excel file that contains the art information.

Format File Name

Choose an existing Format filename if it has been saved previously. Otherwise leave it blank.

Start Row	The first starting row number and the last row number of the parts. Only numeric input is accepted.
End Row	
Mandatory Fields:	Indicate the corresponding Column numbers as in the Excel file for Part Name and Quantity.
Part Name	
Quantity	
Optional Fields:	These are optional fields. Indicate the corresponding Column numbers as in the Excel file for Filler Quantity, Orientation and Priority, if they are available.
Filler Quantity	
Orientation	
Priority	
	Click this button to proceed to TaskEdit .
	Click this icon to view error messages, if any. This is especially helpful as it will give a list of the filenames that cannot be converted for certain reasons.

If you have not defined the Part Information via the button of the same name, when you click “**Next**” a dialog will appear to enable you to enter the quantity of all the converted parts. Default is “1”. After that the **TaskEdit** dialog box will appear.

If any Part has a different quantity, you can change it in **TaskEdit** where you will specify the Stock and other cutting information.

View Part



This provides the facility to view previously created Parts stored in the default directory. If necessary, you can ask for a listing of all the **DWG, DXF and VEC** parts within the directory. As **DWG** parts are stored as AutoCAD's **BLOCKS**, they will be recalled very much the same way as AutoCAD's **INSERT** command.

The following will appear sequentially at the AutoCAD command prompts:

ViewPart

Select part format to view –

Dwg\Dxf\Vec <Vec> :

Insertion Pt:

Part Name (? for list)<>:

Proceed to view another Part? (Y/N) <Y> :

Select part format to view – Dwg\Dxf\Vec <Vec>:

This is to prompt for the type of file format that you wish to view.

- “D” for Dwg
- “X” for Dxf
- “V” for Vec

"Enter" for the default option, in this case the VEC file format.

Part Name (? for list)<>:

This is the prompt for the name of the part, which you wish to review.

Enter the required part name.

Conversely, you can enter **?**, which in turn will display a dialog box for you to select the part name.

Insertion Pt:

Pick a point on the screen. The specified part will be displayed at the insertion point on the screen.

Proceed to view another Part? (Y/N) <Y> :

This prompt will allow you to view another part without quitting the command.

CAUTION:

When the nested layout is displayed in **DWG** format, you may encounter Parts overlapping, nested outside the stocks or parts not updated.

You can avert the problem by starting a new drawing and then view the nested results using **ViewNEST**. There is no need to re-nest.

The reason for this situation is as follows:- Each *AutoNEST* Part saved using **SavePart** will be stored as an AutoCAD block (.DWG) file besides .VEC and .DXF files. AutoCAD will still remember all the Blocks in its memory of the current drawing, regardless whether the Blocks were deleted or not.

This may pose a problem when Parts are edited and then re-saved. If you find that the nested results are not displaying the edited Parts in .DWG format, you can solve the problem by starting a NEW drawing and then **ViewNEST**.

In another situation, where the current drawing has different blocks of the SAME name, a copy of the Block in the current drawing will be inserted. This is all right if they are actually of the same geometry. Otherwise, it can be very confusing. (*AutoNEST* allows SAME names of Parts (**Wblock**) as long as they are in different folders)

This situation can be even more baffling if the existing Blocks have been erased from the current drawing yet AutoCAD remembers them in the drawing! This problem can be eliminated to a certain extent if unused blocks are purged from the drawing (AutoCAD **PURGE** command). This is the only way AutoCAD removes completely unused blocks. Starting a fresh new drawing will also help.

An experienced AutoCAD user will know how to avoid this pitfall.

Save Cluster Part



SaveClusterPart command enables you to save selected two or more *AutoNEST* parts into a cluster.

This command is helpful if the group of parts need to match the GRAIN pattern direction so that all parts would look like the that they came from the same piece of material like lumber.

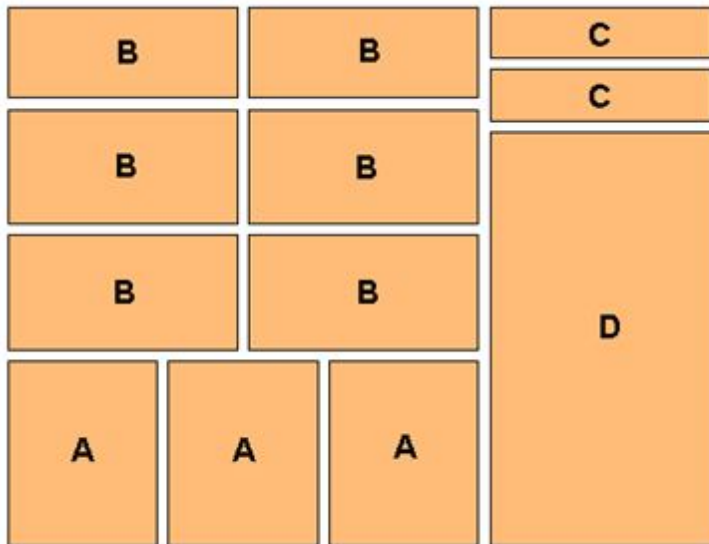


Fig. 1

The grain pattern of Parts A, B, C & D has to match.

SaveClusterPart assumes that the individual *AutoNEST* parts have been created before the command is initiated. *AutoNEST* parts should be arranged in a pattern that you want them to appear on the nested result. Make sure that the parts cutting gap and orientation are correct.

The following prompts will appear at the command prompt when you click **SaveClusterPart** from the *AutoNEST* pulldown or icon menu.

Save Cluster-part:

Cluster Name <> :

Quantity <1> :

Insertion Point:

Indicate each individual AutoNEST Part:

Select individual Part (if there are more than one of the same Part, select them all):

Select objects:

AutoNEST Part Name (? for list) <> : ?

Continue to select individual Part? (Y/N) <Y> :

Window all the parts in the cluster <ENTER> OR indicate the outline of the cluster <O>:

Select objects:

Proceed to save another Cluster-part? (Y/N) <Y> :

Cluster Name <> :

This is to prompt for the name of the new Cluster-part. (SPACE character is not accepted)

File already exists. OK to overwrite? (Y/N): <N>

This prompt will be displayed if the entered Cluster-part name already exists.

Quantity <1> :

Enter the quantity of the Cluster-part to be nested, if the information is available.

Insertion Point:

This is the pick-up point of the Cluster-part as well as the location for label or tag of the Cluster-part.

Select individual Part (if there are more than one of the same Part, select them all):

Select objects:

Pick the individual *AutoNEST* Part to be saved as Cluster-part using any of the available AutoCAD's Entity Select modes. If there are more than one of the same Part, select them all.

AutoNEST Part Name (? for list) <> : ?

Enter “?” to browse or type-in the *AutoNEST* Part name.

Continue to select individual Part? (Y/N) <Y>

This will enable you to select another individual *AutoNEST* part to be saved.

Window all the parts in the cluster <ENTER> OR indicate the outline of the cluster<O>:

Select objects:

Window or select all the *AutoNEST* parts in the cluster or indicate your own Outline.

If you select all the parts, by default an enclosing rectangle is formed. This enclosing rectangle (dashed line) serves as the outline for the Cluster-parts (see Fig. 2). If you were to draw your own outline, ensure that the outline is a continuous polyline or circle.

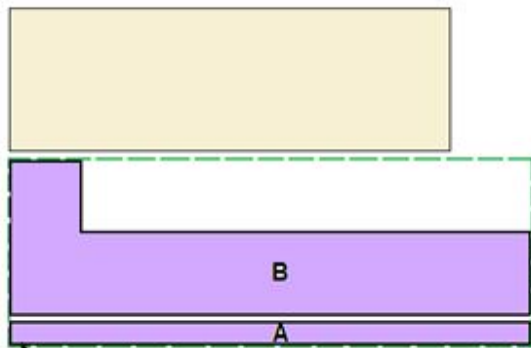


Fig. 2 Nested Cluster-part

Default enclosing rectangle (indicated in dashed line)

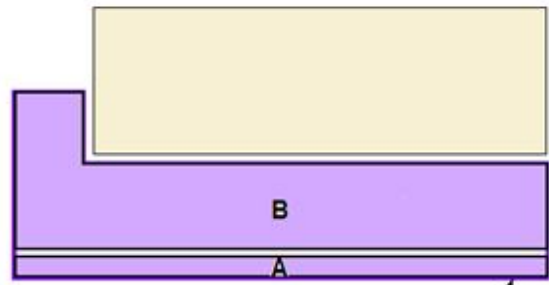


Fig. 3 Nested Cluster-part

User's drawn outline (continuous polyline)

Proceed to save another Cluster-part? (Y/N) <Y> :

This prompt will allow you to save another Cluster-part without quitting the command.

Note: During nesting, no other parts will be nested inside the Cluster-part's enclosing rectangle even though there is available space. In the example above, you are recommended to create your own Outline to maximize material.

In **SaveClusterPart**, the following files are created:-

- **CLS** is a text file which record the relative coordinate of part's insertion point to Cluster-part's insertion point and part rotation-angle.
- **VEC** is a text file which record geometric information for a part, such as x, y coordinates of vertices, bulge values for arcs, center point, coordinates and radius for circles.

View Cluster Part



This provides the facility to view previously saved Cluster-parts stored in the default directory. At the AutoCAD command prompt you will be able to view a Cluster-part in .VEC, .DXG and .DXF file formats.

The following will appear sequentially at the AutoCAD command prompts when you click **ViewClusterPart** from the *AutoNEST* pulldown or icon menu:

Select part format to view –
Dwg\ldXf\Vec <Vec>:

Cluster-part Name (? for list) <> :
Insertion Point:

Proceed to view another Cluster-part? (Y/N) <Y>:

Select part format to view – Dwg\dxflVec <Vec>:

This is to prompt for the type of file format that you wish to view.

- “D” for Dwg
- “X” for Dxf
- “V” for Vec

Cluster-part Name (? for list) <>:

This is to prompt for the name of the Cluster-part, which you wish to review.
Enter the required Cluster-part name.

Conversely, you can enter **?**, which in turn will display a dialog box for you to select the Cluster-part name.

Insertion Point:

Pick a point on the screen. The specified Cluster-part will be displayed at the insertion point on the screen.

Proceed to view another Cluster-part? (Y/N) <Y>:

This prompt will allow you to view another Cluster-part without quitting the command.

Bridge Part (Staydown)



BridgePart enables you to select any number of parts or closed profiles from an AutoNEST layout OR a manually nested drawing and bridge them automatically. It does not matter whether the parts are from AutoNEST or not. What is more important is that those selected parts must be either a **closed polyline** or **circle**.

This command is helpful if you wish to bridge only selected parts (for example, after editing) rather than to bridge all the parts in all the layouts. In that case, **Auto-bridge** (see [TaskEdit](#)) will be able to do very nicely.

The following prompts will appear at the command prompt when you click **BridgePart** from the AutoNEST pulldown or icon menu.

Bridge Part :

Import BRIDGE values from Task file? (Y/N) <N>:

Bridge Width <10.0> :

Min. Corner Offset <20.0> :

Min. Angle <45.0> :

Max. No. of Bridges per Part <5> :

Select adjacent Parts (at least 2):

Please ensure all selected parts are NOT touching or intersecting one another.

Select objects:

To continue ? (Y/N) <N>:

Import BRIDGE values from Task file? (Y/N) <N>:

If you know the AutoNEST TASK name of the nested parts, enter "Y". Otherwise, press <Enter> to assume the default value which is "No".

If "Y" is entered, you will be asked the Task name.

Task Name (? for list):

Enter "?" to browse or type-in the Task name. Next, all the BRIDGE specifications as defined in the TASK will be displayed.

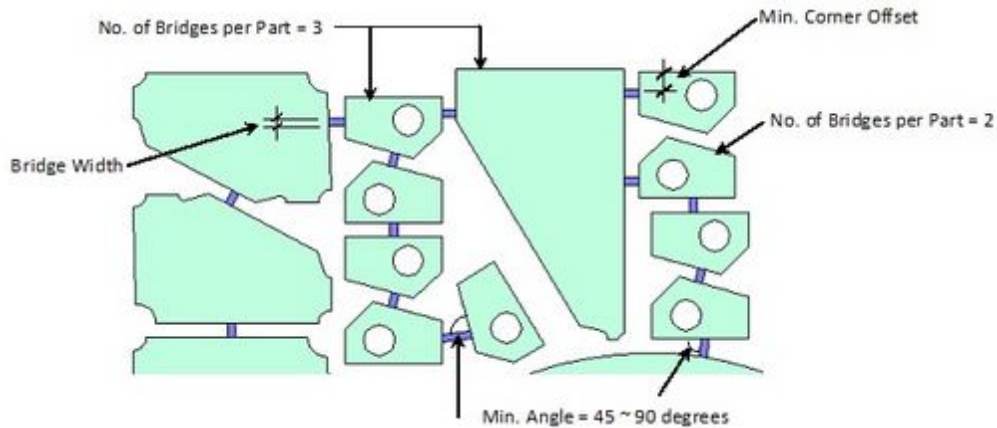
If "N" is entered, then you will be asked ...

Bridge Width <10.0> :

Min. Corner Offset <20.0> :

Min. Angle <45.0> :

Max. No. of Bridges per Part <5> :



Select adjacent Parts (at least 2) :

Please ensure all selected parts are NOT touching or intersecting one another.

Pick the Parts (at least 2) or closed profiles to be bridged using any of the available AutoCAD's Entity Select modes.

NOTE :

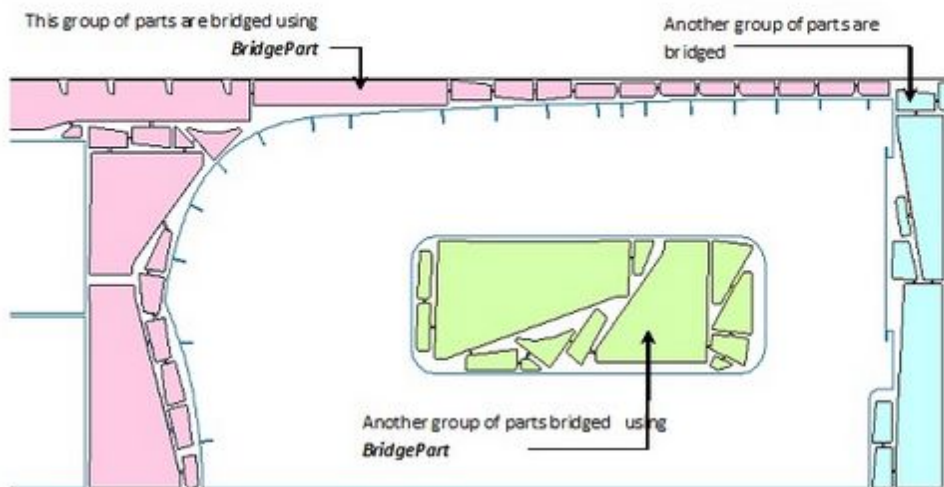
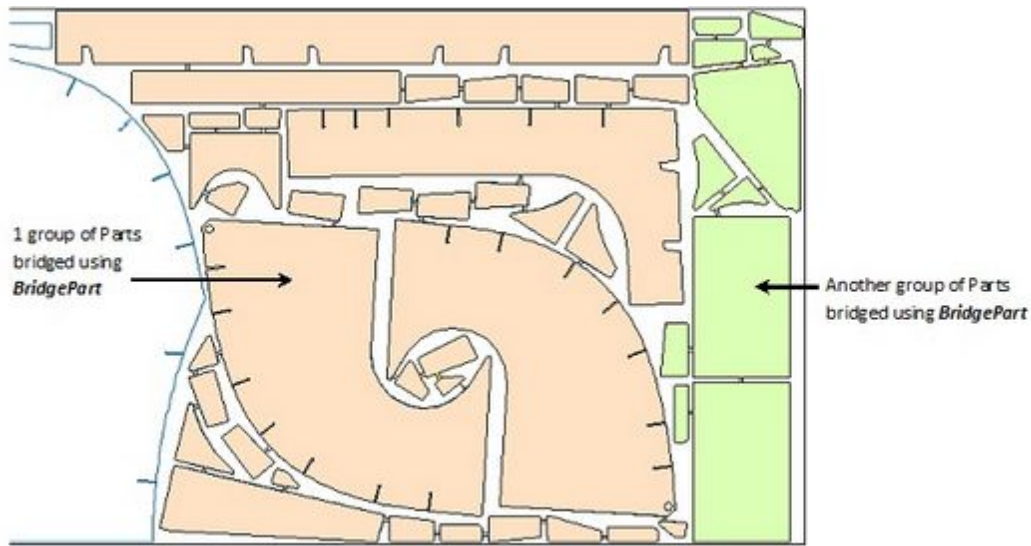
The part profile MUST be a closed profile or circle. Both the external and internal profiles must consist of a continuous polyline or circle. BLOCK (.dwg) will not be accepted.

BridgePart will proceed to bridge the selected parts or profiles. If the results of the auto-bridging is not to your satisfaction, you can redo by removing it and repeating the process. But change the Bridge Specifications the second time round.

To continue ? (Y/N) <N> :

This will enable you to select another group of parts or closed profiles to be bridged.

Press <Enter> to terminate.



Break Part



There are some Parts that are so big that they cannot fit into any one stock. In another situation, there are Parts which by design have to be formed by joining two or more parts together. All these parts have to be “broken-up” into smaller pieces.

BreakPart command will enable you to “break” any selected part into multiple smaller parts. **BreakPart** is quite similar in function to **SavePart** but they are NOT the same.

In **SavePart**, the following files of each Part are created:-

.VEC , .DWG , and .DXF where

- **VEC** is a text file which record geometric information for a part, such as x, y coordinates of vertices, bulge values for arcs, center point, coordinates and radius for circles.
- **DWG** is the drawing file generated by AutoCAD when a part is saved as a **Block /WBlock**. This is to keep a true copy of the part geometry submitted to *AutoNEST*.
- **DXF** is a Drawing Interchange File format to assist in interchanging drawings between AutoCAD and other programs.

In **BreakPart**, the following files of each Part are created:-

- **VEC , DWG & DXF** of the original Part (same as **SavePart**)
- **VEC** of each individual “broken-up” Parts
- **VEC** of the combined “broken-up” Parts

For example, Part Name : ABC

ABC.VEC	}
ABC.DXF	}Original Part (unbroken), same as SavePart
ABC.DWG	}
ABC-1.VEC, ABC-1.dxf	}
ABC-2.VEC, ABC-2.dxf	} .VEC of individual broken-up Parts
ABC-3.VEC, ABC-3.dxf	}
ABC-N.VEC, ABC-N.dxf	}
ABC-\$.VEC **	} .VEC of the combined broken-up Parts
	}This file CANNOT be nested as it contains each and every individual broken-up parts / profiles.

**** IMPORTANT **** This file cannot be nested as it contains more than 1 profile.

When the **BreakPart** command is initiated either from the *AutoNEST* pulldown or icon menu, the following prompts will appear at the AutoCAD command prompt:

Break Part :

Select objects :

Break Part – indicate [Line/Distance/Automatic] <A>:

Break Part - Break into how many parts? <2> :

Part Name <> :

Part Quantity <> :

Require to save holes in Part? (Y/N) <Y> :

X nos. of Parts successfully created.

Select objects:

The Entity Select modes of AutoCAD are applicable here. You can type **W** (Window), **C** (Crossing), **R** (Remove) or simply pick the required entities to be saved as part.

If **W** (Window) is entered at this stage, a dynamic window will be formed to enclose the relevant geometry of the part. All the entities within the window will be saved as an *AutoNEST* part.

Break Part – indicate [Line/Distance/Automatic] <A>:

3 options to break the selected part are available here :-

Enter **L** (Line) to indicate a cutting line interactively on the screen.

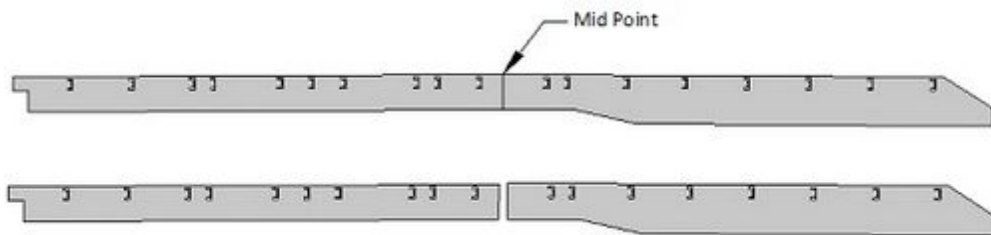
Enter **D** (Distance) to specify a distance from left-most point of the part.

Press **Enter** (default) or **A** (Automatic) to break Part automatically at mid-point of the part.

If **A** (Automatic) is entered, **BreakPart**, the following prompts appear:

Break Part - break into how many parts? <2> :

If “2” is entered, **BreakPart** will automatically break the selected Part at mid-point of the Part. See above. The maximum no. of sub-parts created by this command is six (6).

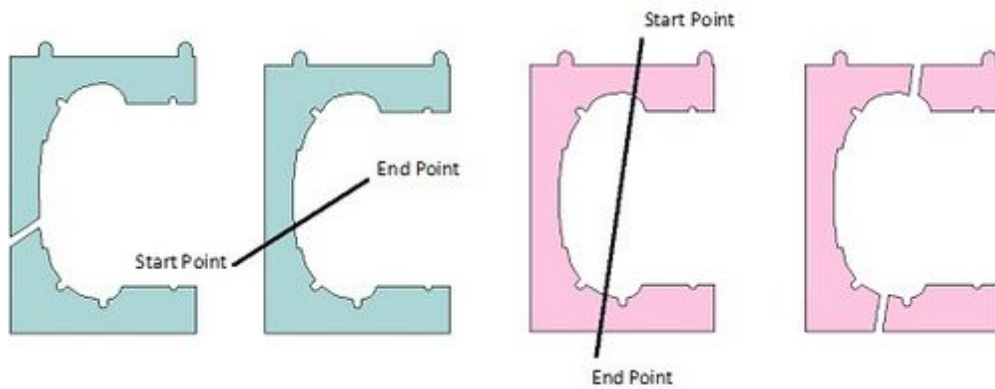


If **L** (Line) is entered, the following prompts will appear :

Break Line – start point :

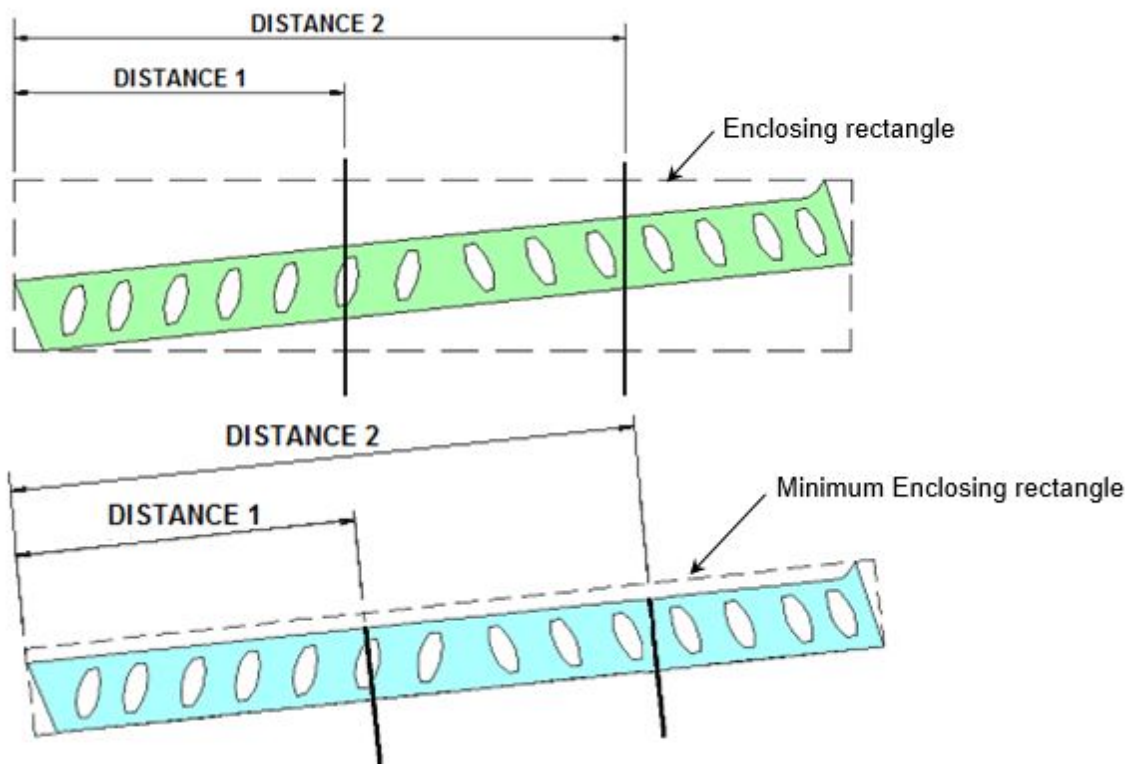
Break Line – end point :

To continue? (Y/N) <Y> :



If **D** (Distance) is entered, the following prompts will appear :

Break Part – reference to [Enclosing/ Minimum enclosing] rectangle <M> :
Distance from Left-hand Side (3000,6000) : (see below)



Part Name <> :

This is to prompt for the name of the new parts. (max. no. of characters allowed : 31, SPACE character is not accepted)

Part Quantity <1> :

Enter the quantity to be nested for this part.

Require to save holes in Part? (Y/N) <Y>:

This will enable the **BreakPart** command to save internal profiles of a part as holes, which may be used to nest other parts.

X nos. of Parts successfully created.

This prompt indicates the number of parts that have been created successfully.

No **Insertion Pt.** is being asked here as the Insertion Point is assumed to be in the middle of the part.

Conditions for Part Profile

Acceptable	Unacceptable
<ul style="list-style-type: none"> Lines, Arcs, Circle, Rectangle, Polyline, Polygon and Spline entities External profile and internal holes profiles of a part must be closed. 	<ul style="list-style-type: none"> Block entity (acceptable if “Explode”) Pedit-Spline and Pedit-Fit (acceptable if “Explode”) Additional line/plines along the profile Crossing over on each profile or between profiles Part with more than 1500 vertices per profile (Inclusive of starting and ending vertices of arcs)

Mirror Part



MirrorPart is basically the same as **SavePart** except in this case, for every part saved, an additional mirrored image of the part will be saved as well.

Likewise, the part has to be drawn first before the **MirrorPart** command can be used.

The following will appear sequentially at the AutoCAD command prompt:

MirrorPart : Part Name <>:

Part Quantity <1>:

Require to save holes in Part? (Y/N)<Y>:

Insertion Point:

Select objects:

Part Name <xxx_M>:

Proceed to mirror another Part? (Y/N) <Y>:

Parts Name <>:

This is to prompt for the name of the part. (Space and dot characters are not accepted).

File already exists. OK to overwrite? (Y/N): <N>

This prompt will be displayed if the entered part name already exists.

Part Quantity <1> :

Enter the quantity to be nested for the parts.

Require to save holes in Part? (Y/N) <Y>:

This will enable the command to save internal profiles of a part as holes, which may be used to nest other parts.

Insertion Point:

This is the pick-up point of the part as well as the location for label or tag of the part. Insertion point is recommended to be one of the vertices of the part or lies within the geometry of the part. AutoCAD object-snap modes under the **OSNAP** command, such as **Endp** or **Midp** are allowed.

Select Objects:

All the Entity Select modes of AutoCAD are applicable here. You can type **W** (Window), **C** (Crossing), **R** (Remove) or simply pick the required entities to be saved as a part.

If W (Window) is entered at this stage, a dynamic window will be formed to enclose the relevant geometry of the part. All the entities within the window will be saved as an *AutoNEST* part.

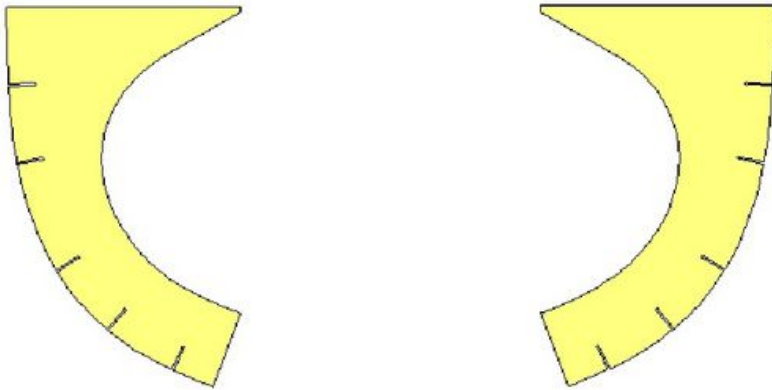
Part Name <TEST1_M>:

By default, the mirror image is given the same name as the original part but suffixed with “_M”. The user has the option to rename it if required.

Proceed to mirror another Part? (Y/N) <Y>:

This prompt will allow you to save and mirror another part without quitting the command.

Example of a part saved by **MirrorPart**, one in its original orientation and the other its mirrored image :



Expand Part



The nested layout displayed after the nesting process in the current AutoCAD session may be edited as a normal AutoCAD drawing. During an editing session of a nested layout, you may wish to make use of a part with an expanded profile by an offset equal to the bridge width. This expanded profile of the part can assist you in manipulating the part to fit a tight corner or space.

When **ExpandPart** is invoked, the following will appear sequentially at the AutoCAD command prompt :

Part Expand:
Insertion Pt:
Part Name (? for list) <>:
Cutting Gap <>:
Proceed to expand another Part? (Y/N) <Y>:

Insertion Pt:

This is to prompt the user the location of the (expanded) part. You can use the pointing device to digitize the screen location or point.

Part Name (? for list)<>

<> will have the name of the most recent entry of part name.

At the '**Part Name (? for list)<>:**' prompt, if the ? option is chosen, the following dialog box is displayed:



Select a Part from the dialog box and click the “**Open**” button.

Cutting Gap:

The part will be expanded by the cutting gap as specified here.

Proceed to expand another Part? (Y/N) <Y>:

This prompt will allow you to continue to “expand” another part without quitting the command.

Once the entries have been confirmed, **ExpandPart** will recall the part with an expanded profile. Again the user can now manipulate the part to fit onto the given stock sheet using AutoCAD **Modify** commands.

The user can then manipulate the part with **MOVE**, **COPY**, **ROTATE**, etc commands of AutoCAD.

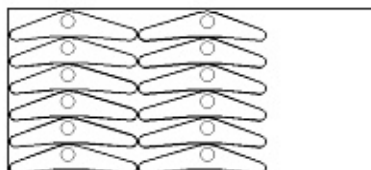
AutoNest Irregular Stock

In AutoNEST, Irregular Stock is a generic name used to describe the following :-

1. Irregular-shaped sheet for nesting
2. Layout with pre-nested parts
3. Remnant sheet /stock



Examples of Irregular-shaped sheet / stock






Example of Layout with pre-nested parts



Example of Remnant sheet / stock

This section introduces three commands related to **Irregular Stock** within the AutoCAD environment. They are used to save or view irregular stocks that have been drawn or edited using AutoCAD commands. In addition, .DXF files can be converted into irregular stocks as well.

Irregular Stocks will also be saved automatically when you marked the “**Save Remnant**” check-box in the **Nest Options** within the **TaskEdit** dialog box. Please refer to [TaskEdit](#). for details.

	SaveIrStock To save an irregular stock which you have constructed or drawn using AutoCAD <i>DRAW</i> commands
	ConvertIrStock To convert DXF files to irregular stocks (.STK). As well as from irregular stock (.STK) files to DXF format.
	ViewIrStock To view a saved irregular stock (.STK file), click this icon.

Save IR Stock



Similarly to Parts, an **Irregular Stock** can be constructed or drawn using AutoCAD **DRAW** commands. The **SaveIrStock** command can be used to save the irregular stock profiles into the Part / IR-Stock directory as specified in [Sysdata](#).

When the command is selected either from the pulldown or the icon menu, the following prompts will appear sequentially at the command prompt:

Save Irregular Stock:
Irregular Stock Name <>:
Quantity <1>:
Insertion Point:
Select object:
Proceed to save another Ir-Stock? (Y/N) <Y>:

Irregular Stock Name <>:

This is the prompt for the name of the new irregular stock. (Max. no. of characters - 31, both the space and dot characters are not accepted)

File already exists. OK to overwrite? (Y/N): <N>

This prompt will be displayed if the entered stock name already exists

Quantity <1> :

Enter the quantity of the irregular stock to be nested.

Insertion Pt:

This is the pick-up point of the stock. Insertion point is recommended to be one of the vertices of the profile or anywhere inside the profile. AutoCAD object-snap modes under the **OSNAP** command, such as **Endp** or **Midp** are allowed.

Select Objects:

The Entity Select modes of AutoCAD are applicable here. You can type **W** (Window), **C** (Crossing), **R** (Remove) or simply pick the required entities to be saved as irregular stock.

If **W** (Window) is entered at this stage, a dynamic window will be formed to enclose the relevant geometry of the stock. All the entities within the window will be saved as an *AutoNEST* irregular stock.

During **SaveIrStock**, the following happens:

- a) **.STK**, **.DWG**, and **.DXF** files are created, where :
 - **STK** is a text file which record geometric information of an irregular stock, such as x, y coordinates of vertices, bulge values for arcs, center point, coordinates and radius for circles.
 - **DWG** is the file format given by AutoCAD when an irregular stock is saved as a **Block/WBlock**. This is to keep a true copy of the geometry submitted to *AutoNEST*.
 - **DXF** is a Drawing Interchange File format to assist in interchanging drawings between AutoCAD and other programs.
- b) Text and dimensioning information are filtered out in the **.STK** file.

Conditions for saving Irregular Stock profile:

Acceptable	Unacceptable
<ul style="list-style-type: none"> Lines, Arcs, Circle, Rectangle, Polyline, Polygon and Spline entities External profile and internal holes profiles of irregular stock must be closed. 	<ul style="list-style-type: none"> Block entity (acceptable if "Explode") Pedit-Spline and Pedit-Fit (acceptable if "Explode") Additional line/plines along the profile Crossing over on each profile or between profiles Part with more than 1500 vertices per profile.

Convert IR Stock

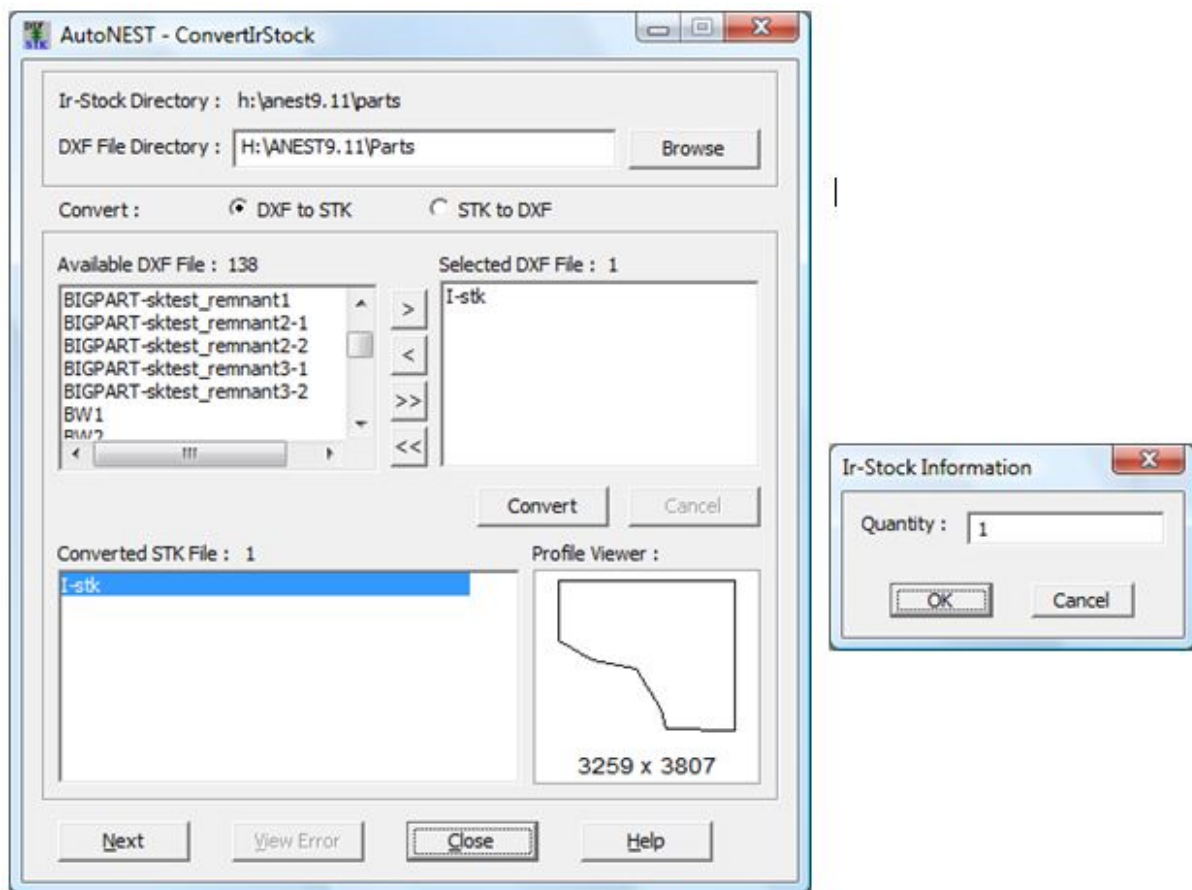


ConvertIrStock is a function that allows the user to convert the irregular stock profiles in *DXF* file format to irregular stocks (*.STK* file format).

Similarly, it allows user to convert irregular stocks (*.STK* file format) back into *DXF* format.

Select the **ConvertIrStock** command icon from the icon menu within the AutoCAD drawing session.

The following dialog box will appear:

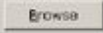


Each dialog box input option is described in the following listing:

IR-Stock Directory Display the default irregular stock directory set at **Sysdata**.

The irregular stock directory is the same as the Parts directory. In this case, it is also the target directory where all converted .stk files will be saved.

DXF File Directory

To type in or click on the  button to choose a directory where the source file are located.



Convert DXF to STK

Select this radio button to convert DXF files to STK files.



Convert STK to DXF


Select this radio button to convert STK files to DXF files.

Available DXF/STK File

The available files in the directory. (This will depend on which option has been chosen). You can select one or more files to be converted by highlighting the filenames and then clicking the  button. By clicking the  button, all the files in the directory will be selected.

Selected DXF/STK File

This list box displays the selected files for the conversion. You can unselect the filenames by clicking the  button to remove the files from the “Selected Parts” list (but the files still remain on the “Available Parts” list) or click the  button to remove all files.

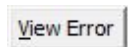
Click the  button to initiate conversion process.

Ignore Holes and Cut-outs

If this check-box is marked, holes and cut-outs will not be converted.

Converted (STK/DXF) File

This will show files that have been converted successfully.



Click this icon to view error messages, if any. This is especially helpful as it will give a list of the filenames that cannot be converted for certain reasons.

The next dialog “**Ir-Stock Information**” will appear. Enter the quantity for all the converted irregular stocks. Default is “1”. If any of the stocks has a different quantity, you can change it in **TaskEdit**. Click the “**Next**” button will bring you to **TaskEdit** where you will specify the Part and other cutting information.

View IR Stock



This provides the facility to view previously created irregular stocks stored in the default directory. If necessary, you can ask for a listing of all the file names within the directory.

The following will appear sequentially at the AutoCAD command prompts:

The screen prompts are:

View Irregular stock (.STK):

Irregular stock name (? for list) <>:

Insertion Pt :

Proceed to view another Ir-Stock? (Y/N) <Y>:

Irregular stock name (? for list)<>:

This is to prompt for the name of an irregular stock in .STK file format that you wish to view. Enter the required stock name.

If you enter “?” at the “**Irregular stock Name (? for list)<>:**” prompt, the following dialog box will be displayed. In this case, you can have access to all the .stk files in the default directory.



Insertion Pt:

Pick a point on the screen. The specified STK stock will be displayed at the insertion point on the screen.

Proceed to view another Ir-Stock? (Y/N) <Y>:

This prompt will allow you to view another irregular stock without quitting the command.

AutoNest Task Edit



In **TaskEdit**, three (3) categories of information are required:

- Stock sheet (regular and irregular stocks accepted)
- Parts to be nested
- Cutting specifications

- Bridge specifications (optional)

Stock sheet information includes the number of distinct stocks, size and quantity of each distinct stock. AutoNEST can handle the following types of stocks:-

- rectangular stocks of fixed size
- irregular-shaped stock sheets
- "coil" stock in rolls or reels.

Information relating to the number of parts, the quantity to be nested for each part and orientation constraint ...etc will also be defined in **TaskEdit**. Cutting information such as cutting gap, x-y edge allowances are also definable.

In addition, you can define the Bridge specifications for the particular task. (A "BRIDGE" is a narrow strip that links one part to another in order to maintain continuous cutting. This minimizes stop/go sequences that would lead to higher machine efficiency)

You can also mark the check-box "Auto-Bridge for Nested layout" to indicate that you wish to display the nested layout with bridges.

There is also a "**Nest Now**" button within the dialog box for nesting. Instead of clicking the **Nesting** command, the user can create/ edit Tasks and go direct into nesting.

Select the **TaskEdit** command either from the icon menu or the AutoNEST pulldown menu. The following dialog box will appear:

AutoNEST - TaskEdit

Task Directory : c:\anest9.11.5()\task

Task Name :

Cutting Gap :

Edge Allowance

Regular Stock / Remnant :

Left : Top :

Right : Bottom :

Irregular Stock / Remnant :

No. of Distinct Regular Stocks : 1

No. of Distinct Irregular Stocks : 0

No. of Distinct Parts : 8

Nesting Control Options :

☐ Auto-Bridge for Nested Layout

☐ Skeleton-Cut-Off ☐ Scrap-Pocket

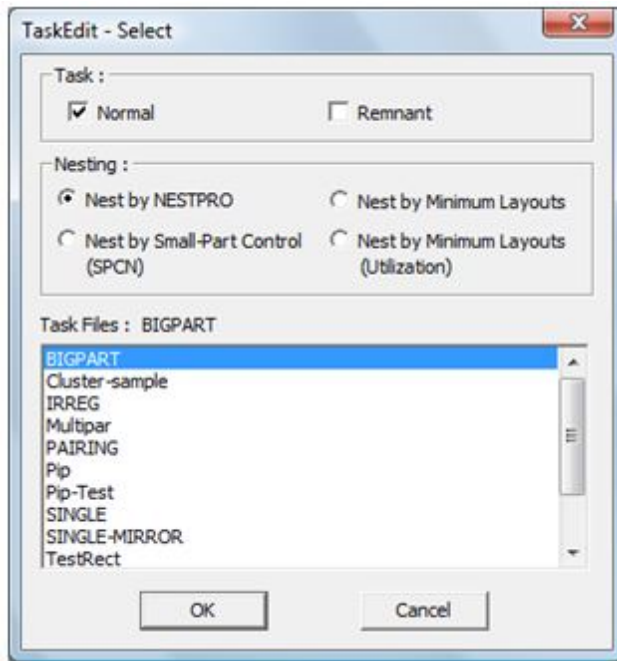
'A'**'B'****'C'****'D'****'E'****'F'**

Explanation of the **TaskEdit** dialog box is given below:

Task Name

Each task is given a user-specified name for future reference (.job). You can enter up to max. 31 characters (space and dot characters are not accepted).

Enter a new task name or click to select an existing task. You will see the Pop-Up Window below.



Task Type

Referring to the pop-up windows below, there are 2 selections :-

- Normal
- Remnant

'Normal' tasks refer to all task files (*.job) created by the user.

'Remnant' tasks refer to tasks files (*.job) with the reserved suffix, '_\$REM' . For example : Task1_REM.job.

These tasks are created automatically when the '**Save Remnant**' check-box is marked in the NEST OPTIONS of **TaskEdit** ("**Nest Options**" button is located at the lower-right corner of the dialog box).

Nesting Type

From version 9.12.2 or higher, there are 4 nesting engines :-

- Nest by NESTPRO
- Nest by Minimum Layouts
- Nest by Minimum Layouts (Utilization)
- Nest by Small-Part Control (SPCN)

You can select the nesting engine of your choice. Detailed explanation of each engine is explained in Chapter 5.5 Nesting.

The default is "**Nest by NESTPRO**" which is recommended to all nesting jobs.

However when the parts' quantities are high, it is recommended you use "**Minimum Layouts (Utilization)**" nesting engine. This engine is designed to optimize the utilization of the stocks as well as to achieve a high repeatability of the layouts, thus minimizing the number of different nested layouts.

"**Minimum Layouts**" engine is only suitable when the stock materials are inexpensive and the objective is the maximize the repeatability of the nested layouts or cutting plan.

The 4th nesting engine – Small-Part Control is particularly suited for cutting machines that has areas of weak suction. Using this nesting engine, “Small-Parts” would not be packed where the suction is weak.

Cutting Gap

This is the cutting gap between nested parts to allow for the tool size.

Edge Allowance -Regular Stock -Irregular Stock

Sometimes, it is necessary to leave out an edge around the perimeter of the stock sheet. This is to cater for trimming or clamping purposes.

For regular or rectangular stock, the edge allowance of the 4 sides of the stock can be defined.

For irregular stock, a common edge allowance can also be defined.

- Left "Left" is the edge allowance of the regular stock in the x- direction from the left side of stock sheet.
- Bottom "Bottom" is the edge allowance of the regular stock in the y- direction from the bottom of the stock sheet.
- Right "Right" is the edge allowance in the x direction on the right side of the stock sheet.
- Top "Top" is the edge allowance in the y direction from the top of the stock sheet.

No. of Distinct Regular Stocks

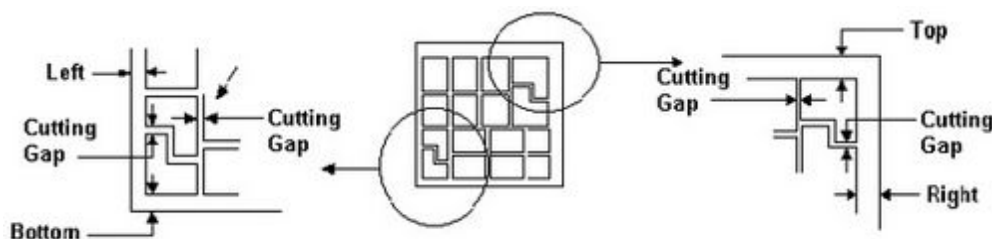
Number of different sizes of stock sheets. (The maximum number of distinct regular stocks can be defined in ANEST.SYS under NEST@CONTROL)

No. of Distinct Irregular Stocks

Number of different sizes of irregular stock sheets. (The maximum number of distinct irregular stocks can be defined in ANEST.SYS under NEST@CONTROL)

No. of Distinct Parts

Number of distinct parts to be nested. A maximum of 1000 distinct parts is available for the current version.



Illustrations for Cutting Gap & Edge Allowance

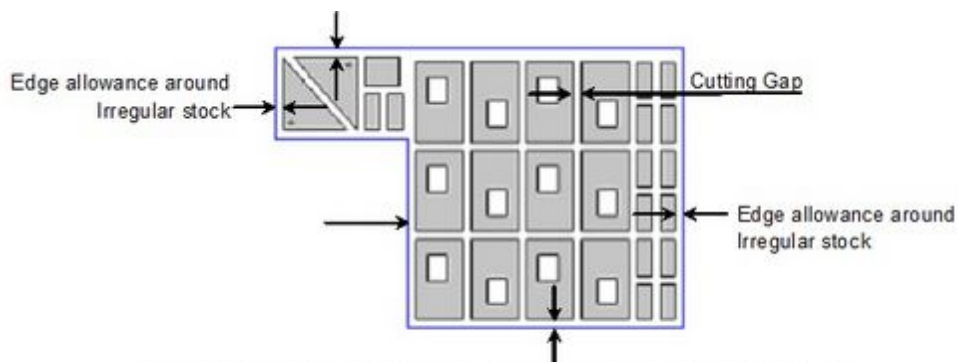
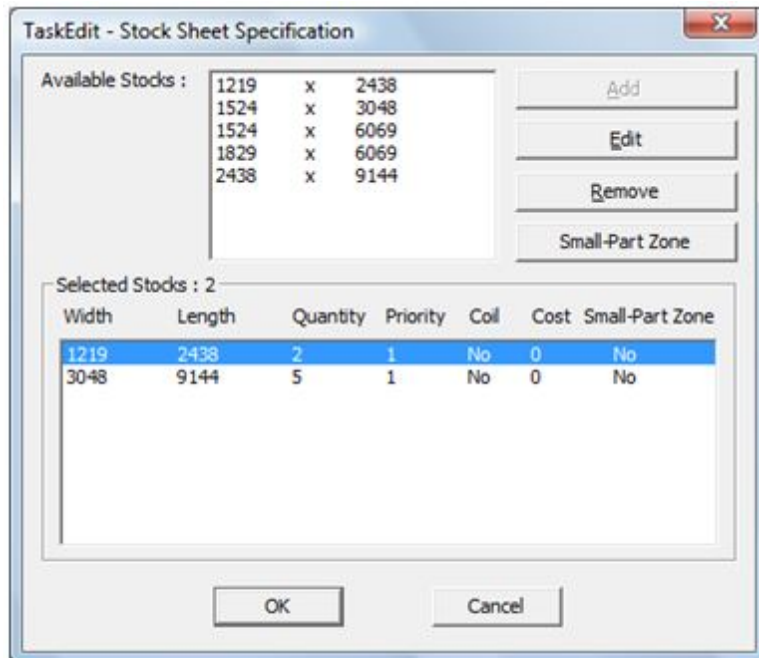


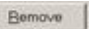
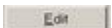



Illustration for Cutting Gap & Edge Allowance of Irregular Stock

Define R-Stocks 'A'

When you click the  button, the following dialog box appears.



You can add, change or delete the stock specification by clicking the ,  or  button respectively. Highlight one of the stock sizes in the "Available Stocks" list box and then pick the  button or click the  button.

The following pop-up windows will appear:

**Width**

The width and length dimensions of the stock sheet.

Length

For 'Imperial Units' users, you can either enter in feet and inches or purely in inches. In the latter case, the inches will automatically be converted into feet and inches if Architectural-Imperial or Engineering-Imperial units are chosen.

Quantity

Quantity of stock-sheets of that particular size to be used in the task. Maximum quantity allowed for each distinct stock is 9999.

Coil

For stock sheets in reels or rolls, the length of the stock sheet can be extended beyond the specified dimension to accommodate the nesting results.

If the '**Coil**' check-box is marked, *AutoNEST* will have the liberty to increase the Length as much as it takes to pack all the parts optimally.

Priority

The priority of using the stock sheets when different stock sizes are available. You can define the order of priority in which *AutoNEST* should use the stock sheets.

1 has the highest priority.

99 has the lowest priority.

If two or more stock sizes have the same priority number, the system will automatically decide based on the material utilization.

To specify the PRIORITY between regular and irregular stocks, '[Nest Options](#)' → 'Nest Regular Stock First'.

Cost per Stock

The cost per piece of Stock.

Small-Part Zone

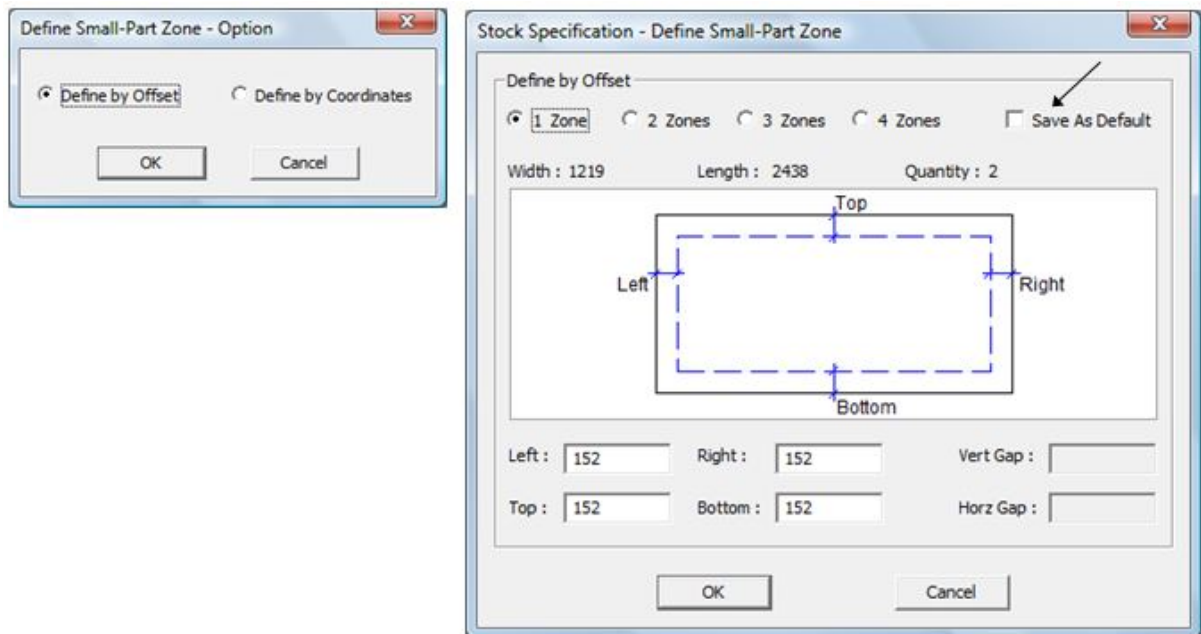
Mark this checkbox to indicate that particular stock has 'Small-Part Zone(s)'.

Small-Part Zone is the area of the stock that has adequate and strong suction, therefore "Small-Parts" should be nested within this zones(s) and not outside.

To define 'Small-Part Zone' on the stock, highlight one of the selected stock and click the




button. The following dialog boxes will be displayed.

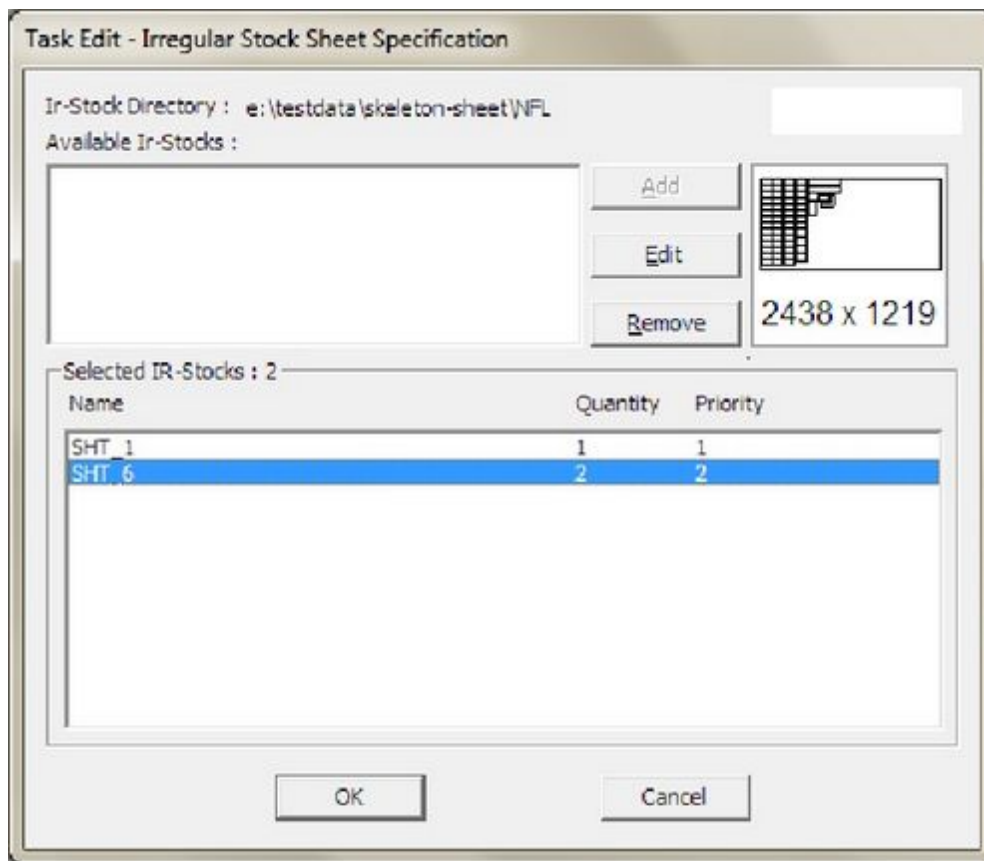







You can define up to a maximum of four (4) small-part zones per stock. You can define using **Offset** dimensions from the stock or by **Coordinates**.

You can also save the 'Small-Part Zone' of a particular stock size by marking the "**Save As Default**" checkbox.

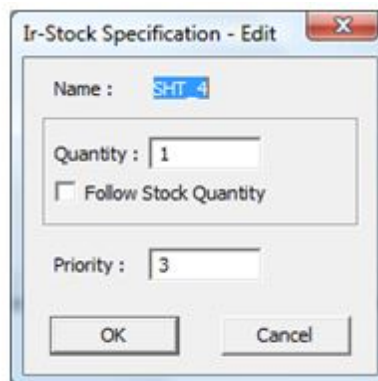
Define IR-Stocks **'B'**

By clicking the  button, the dialog box below appears.



You can add, change or delete the irregular stock specification by clicking the ,  or  button respectively. You can highlight more than 1 stock name and then select the  button or click the  button.

The following pop-up window will appear:



Name

Show the name of the Irregular stock (.stk). If a list of irregular stock is selected, “\$\$\$\$\$\$\$” will appear.

Quantity

Quantity of that particular irregular stock to be used in the task.

Maximum quantity allowed for each distinct stock is 9999.

Priority

The priority of using the stock sheets when more than one is available. You can define the order of priority in which *AutoNEST* should use the Irregular stock.

1 has the highest priority.

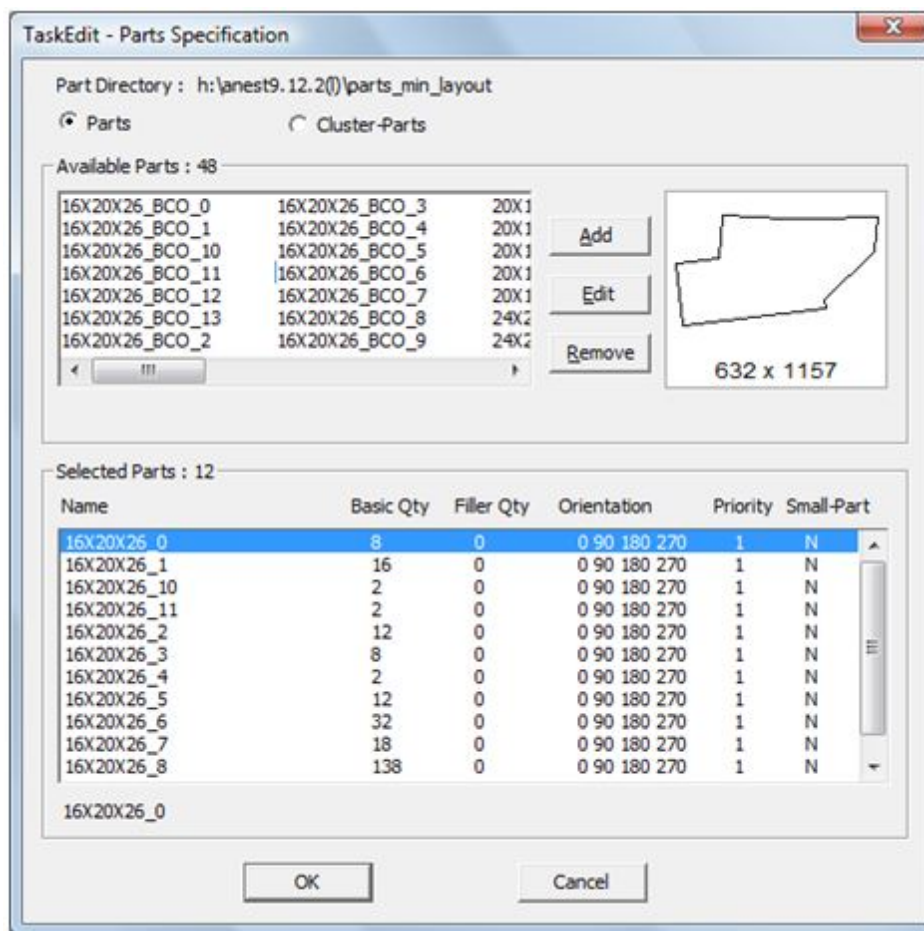
99 has the lowest priority.

If two or more stocks have the same priority number, the system will nest the smallest Irregular stock first.

To specify the PRIORITY between regular and irregular stocks, see '[Nest Options](#)' → 'Nest Regular Stock First'.

Define Parts 'C'


By clicking the  button, the dialog box below appears:

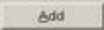


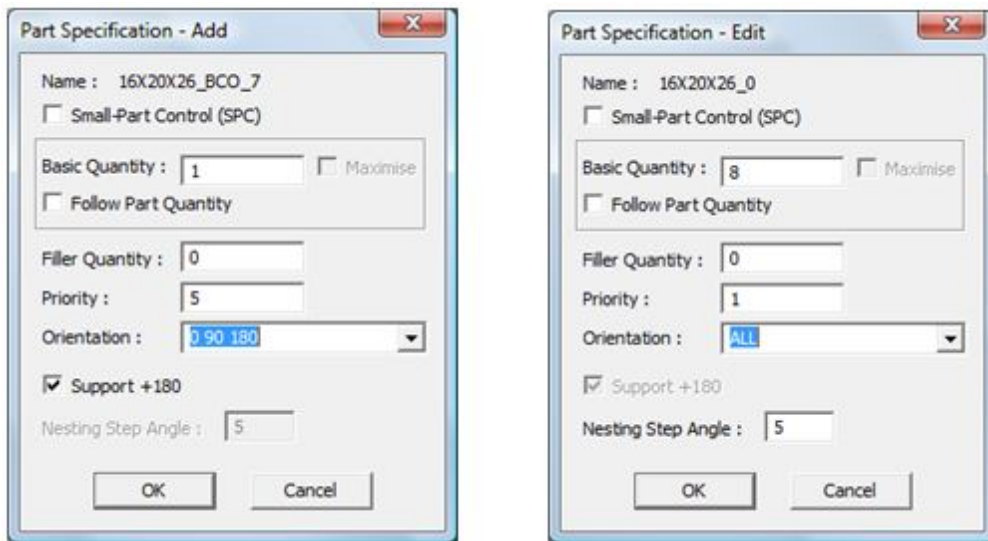
There are 2 radio-buttons :- "**Parts**" and "**Cluster-Parts**". Depending on which radio-button is selected, all the parts or cluster-parts on the Parts directory will be shown in the '**Available Parts**' list box.

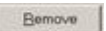

Whenever a part name from the listing of '**Available Parts**' is highlighted, a viewer will display the part profile with the overall dimension of the part profile. The overall length and breadth dimensions will be rounded off to the nearest integer value.

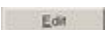
Available Parts

Display a list of parts that are available in the part directory. You can highlight a list of parts and click the  button to add them to the '**Selected Parts**' list box.

The following pop-up window appears when the  button is clicked:



Both the **Add** and **Edit** pop-up windows allow editing. The  button will remove the part specification from the '**Selected Parts**' list box. The  button will place it back to the '**Selected Parts**' list.

If you select the  button the above pop-up windows will appear.

Explanation of the above dialog boxes is given below:

Name	Show the name of the part that you have selected. If more than 1 part is selected, '\$\$\$\$\$\$\$' will appear.
Small-Part Control (SPC)	Mark this checkbox to indicate that the part is a "Small-Part". This information will be used when the "Small-Part Control Nesting" or SPCN is being run.
Basic Quantity	Number of that particular part required to be nested. Each part has a maximum quantity of 9999.
Maximize	For single part tasks, mark the ' Maximize ' check-box to specify 'fill-up' the whole stock sheet with the maximum quantity of parts. Not valid for coil stocks.

Follow Part Quantity

This checkbox is marked when the part quantity has already been defined. But you can still unmark it if you wish to change the quantity of the part.

Filler Quantity

Numbers of that particular filler part to be nested. Each filler part has a maximum quantity of 9999.

This is 'optional' quantity, to be used provided all the basic quantities have been nested and there is available space. AutoNEST will decide how many filler parts to be utilized in order to fill the stock to an acceptable layout.

Priority

The priority of each and every part to be nested.

1 has the highest priority.

9999 has the lowest priority.

AutoNEST will nest according to the priority settings i.e. parts of Priority 1 will be nested first, followed by Priority 2, then 3 and so on. If two or more parts have the same priority, AutoNEST will decide automatically which part is to be nested first.

Orientation

To allow for orientation constraints of the part during nesting.

Format A

0 -- No rotation allowed.

0 90 -- 0° and 90° orientations allowed.

0 180 -- 0° and 180° orientations allowed.

0 90 180 -- 0°, 90° and 180° orientations allowed.

ALL -- All orientations allowed. No orientation constraints.

Besides the above allowable orientations, you can specify any required angles of orientation (separated by SPACE). For examples :-

0 10 20 30 40 50 60 70 80 90

Format B

This new format is helpful if the Orientation angles consist of a long list and if the angles increase by a constant value.

(step 10 0 180)

where:

step is the reserved word to denote "increment angle"

10 is the increment angle (must be greater than 0 and a positive value)

0 is the Start Angle (real nos., range -360 ~ +360)

180 is the End Angle (real nos., must be greater than Start Angle)

If represented in Format A

(0 10 20 30 40 50 60 70 80 90 100 110 120 130 140.....180)

Support + 180


Mark the check-box, to allow the part to orientate to the specified angles and their +180° combinations.

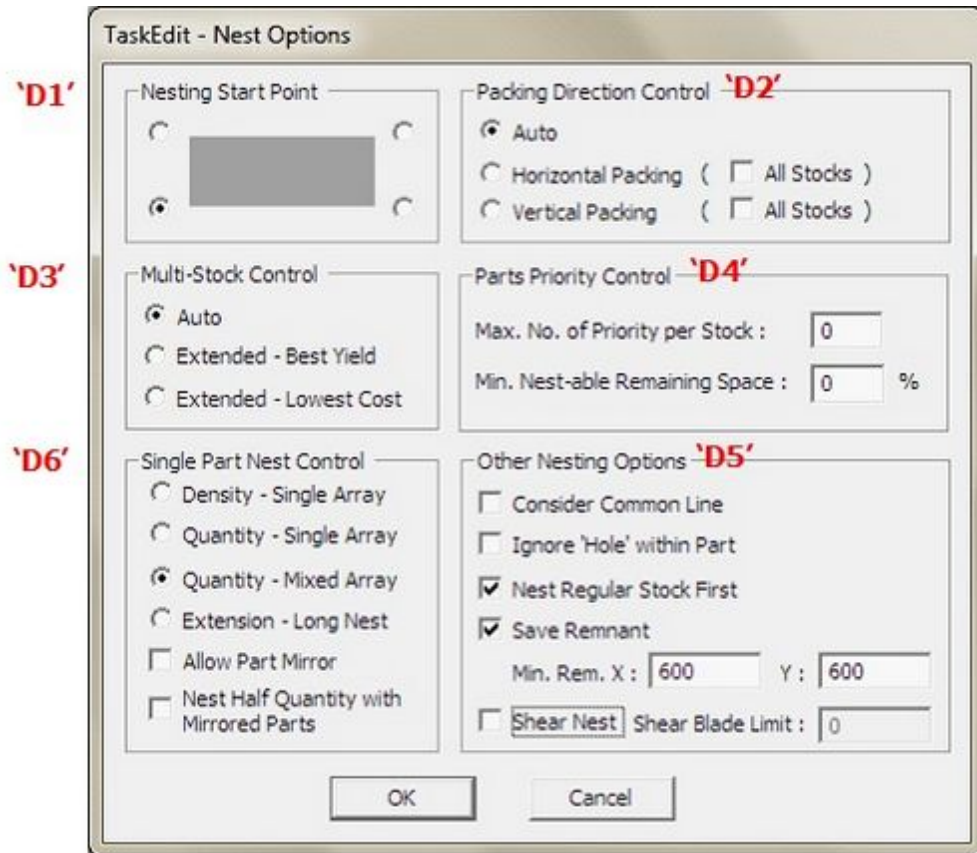
This option is not applicable when **Orientation=ALL**.

Nesting Step Angle

To allow a part to orientate step-incrementally in the defined angle during nesting.

Nest Options 'D'

Click the  command button to bring up the dialog box as shown:



This dialog box allows the user to define control parameters for specific nesting output requirements.

The default settings are recommended for all users who do not require special nesting output conditions.

The options available are explained as follows:

'D1'

Nesting Start Point 4 nesting start points are available depending on the user's nested output requirements. Click one of the 4 corner points to set start point. Point chosen shall remain as default until it is changed.

'D2'

Packing Direction Control

It is recommended for the packing direction to be set to '**Auto**'. However, if required either "Horizontal Packing" (pack along the x-axis) or "vertical packing (pack along the y-axis) can be selected.

Auto AutoNEST will automatically decide the best packing direction based on the material utilization of the nesting. This is the default setting. See illustration below.

Horizontal Packing When this radio button is marked, only the last nested layout will be packed horizontally.

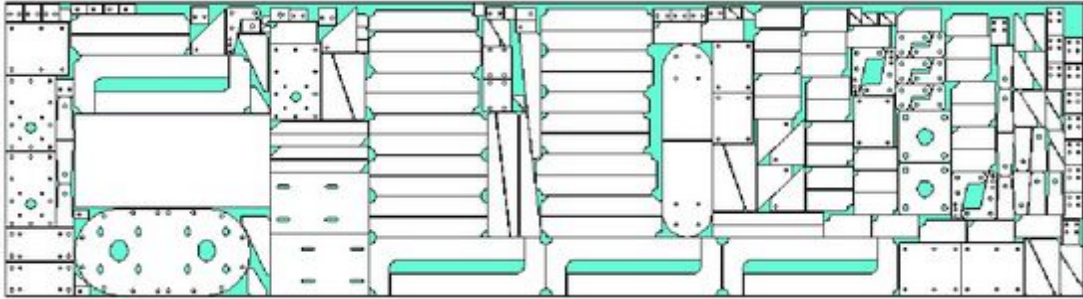
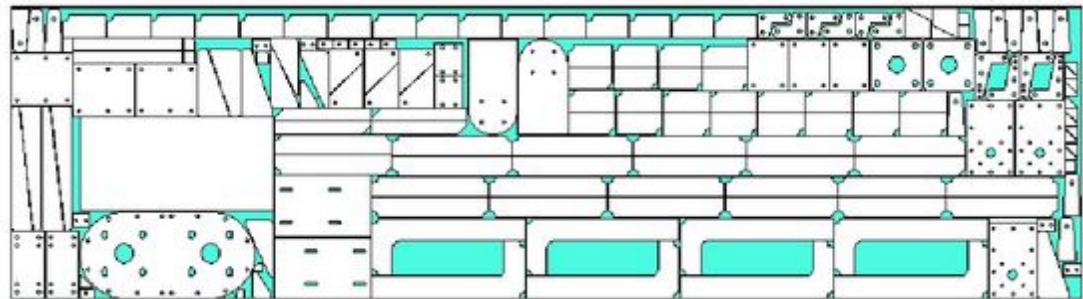
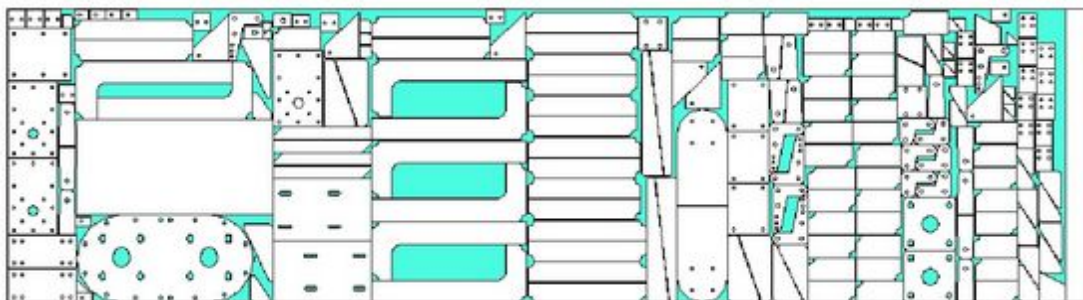


However, when the "**All Stocks**" check-box is marked, then all the nested layouts will be packed horizontally. See illustration below.

Vertical Packing When this radio button is marked, only the last nested layout will be packed vertically.



However, when the "**All Stocks**" check-box is marked, then all the nested layouts will be packed vertically. See illustration below.

Auto Packing:**Horizontal Packing:****Vertical Packing:****'D3'****Multi-Stock
Control**

There are 3 options available.

- **Auto**
- **Extended – Best Yield**
- **Extended – Lowest Cost**

“**Auto**” is the default setting and is the fastest option where the nesting engine will appraise the parts and stocks and then automatically selects the largest suitable Stock to nest.

“**Best Yield**” – in this option the nesting engine will iterate through each Stock size to nest the available parts. For each nested Stock, it will compare the nested utilization area of Stock and then select the one with the highest utilization. It will continue to iterate until all the parts or stocks are used up.

“**Lowest Cost**” – in this option the nesting engine will iterate through each Stock size to nest the available parts. For each nested Stock, it will compare

the Nested Area per Cost of Stock and then select the one with the highest Nested Area per unit cost.

For this option to work, the “Cost per Stock” must be specified.

Both the “Best Yield” and “Lowest Cost” will take a longer time to nest.

*****The more the number of distinct Stocks, the longer it will take to nest for “Best Yield” or “Lowest Cost” options*****

If either “Best Yield” or “Lowest Cost” option is selected, the Stock Priority will be ignored.

‘D4’

Parts Priority Control

This function works in conjunction with “Priority” of parts as defined in “Define Parts” ‘C’.

There are 2 Parts Priority parameters to enable you to control the nesting of parts.

- **Max. No. of Priority per Stock**
- **Min. Nest-able Remaining Space (%)**

Max. No. of Priority per Stock “**Max. No. of Priority per Stock**” enables you to limit the no. of Parts of different priorities to be nested into a stock.

This is to avoid the confusion on the production floor where the operator, after cutting has to sort the parts and placed them into their respective “carts”.

Min. Nest-able Remaining Space % “**Minimum Nest-able Remaining Space**” **percentage** – is the parameter for the remaining space after ALL the parts of the specified Priority have been nested onto the stock.

If **0** is defined, then any remaining space is considered “nest-able”, AutoNEST will try to find suitable parts to fit onto the remaining space.

If **10** is defined, the remaining space must be $\geq 10\%$ of the stock size to be considered “nest-able”, AutoNEST will then try to find suitable parts to fit onto the remaining space.

If **100** is defined, then any remaining space is considered NOT “nest-able”, AutoNEST will not attempt to nest into the remaining space after ALL parts of the specified priority have been nested.

‘D5’

Other Nesting Options

The nesting options are:

- **Consider Common Line**
- **Ignore ‘Hole’ within Part**
- **Nest Regular Stock First.**
- **Save Remnant**

- **Min. Remnant X / Y**

Consider Common Line



Select this option if you want *AutoNEST* to consider packing of the parts along their common edges. This feature will save cutting time.

However, *AutoNEST* will consider the common line packing together with the quality of the nesting.

Ignore 'Hole' within Part

Mark this check-box to ignore all "holes" and cut outs within Parts during nesting. So that no parts will be packed into these "holes" or cut-outs.

Nest Regular Stock First

Mark this check-box to nest regular stocks first. The default is to nest irregular stocks first.

Save Remnant

Mark this check-box to save the remnants of the stocks automatically after every nest, as long as they are of the minimum X and Y size or more. By default, the original profile of the rectangular stocks will also be saved with the remnants. However this can be changed in ANEST.SYS @REMNANT_CONTROL.

By default these remnants will be saved into the Part/ Irregular Stock directory. The names of the remnant stocks will be *Taskname_\$REM1-1.stk*, *Taskname_\$REM2-1.stk*, *Taskname_\$REM2-2.stk* and so on.

The naming convention is as follows :- *_\$REM(LAYOUT_NO)-(NO) where the suffix "\$REM" can be defined in @REMNANT_CONTROL of ANEST.SYS.

In addition, a Task will automatically be created bearing the name, *Taskname_\$REM.job*.

Min.Rem.X Min.Rem.Y

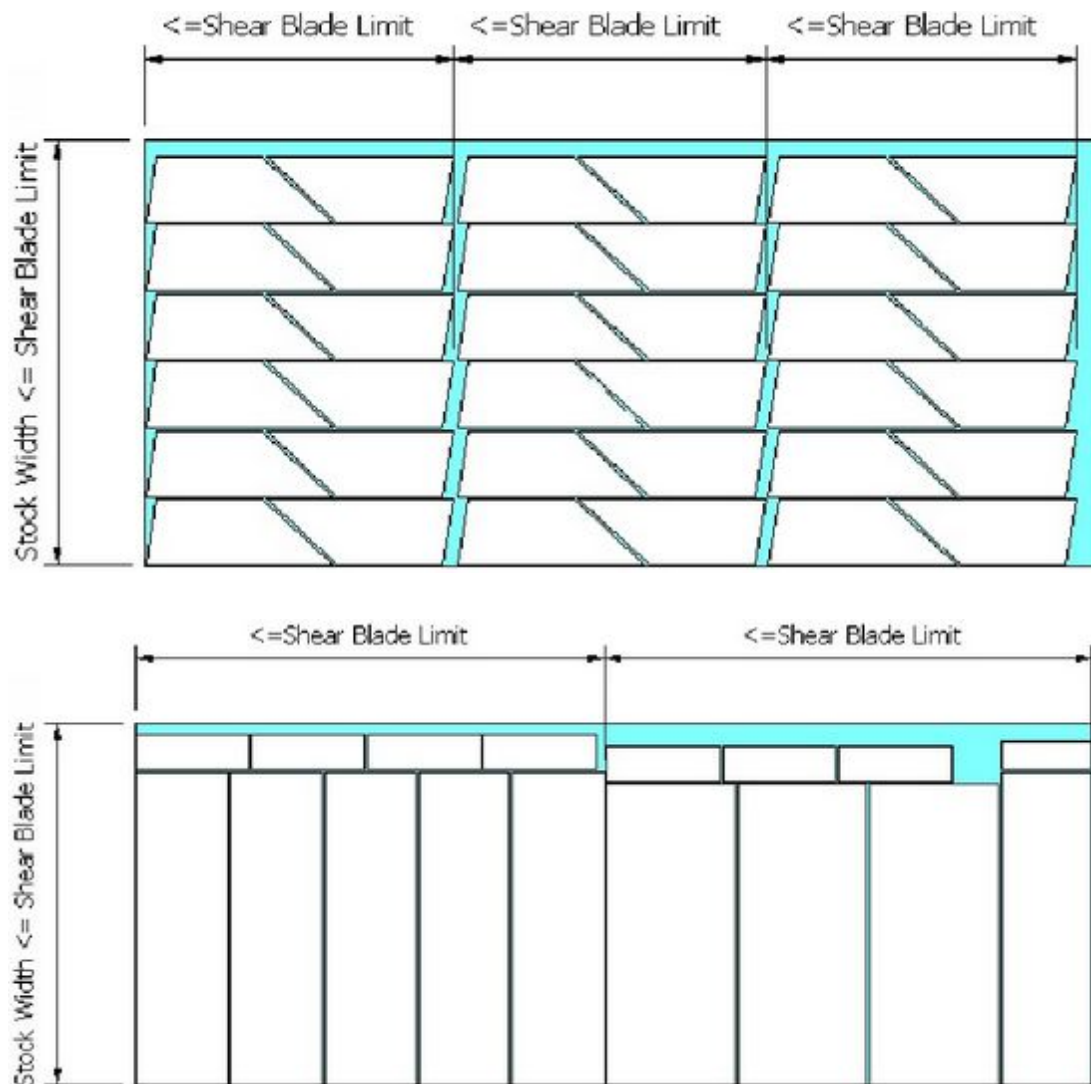
The minimum remnant size as in the X and Y dimensions. The default dimension is 600 x 600. The maximum value is limited to 6 digits (Eg. 999999).

Shear Nest

Mark this check-box to indicate "Shear Nesting" for that particular task.

Shear Blade Limit

When the 'Shear Nest' check-box is marked, you will be able to enter the 'Shear Blade Limit'. See below.

**'D6'****Single Part Nest Control**

This portion of the dialog box will be grey-out unless the Task consists of only ONE Part.

For Single-part nesting, there are **6** options or controls. They are:-

- **Density - Single array**
- **Quantity - Single array**
- **Quantity - Mixed array**
- **Extension - Long Nest**
- **Allow Part Mirror**
- **Nest Half Quantity with Mirrored parts**

The illustrations will give you a better understanding of these control settings

Density-Single array

The most-dense packing pattern in a single array.

Quantity-Single array

The highest quantity of parts nested in a single array.

Quantity-Mixed array

This is the default option. Maximum quantity of parts nested onto the stock sheet.

Extension-Long Nest

This nesting option will take the longest time compared with the other three. It will go through more iteration to produce either the same or better results than Quantity-Mixed array.

Allow Part Mirror

If this option is checked, the nesting will consider the part (single) and its mirrored image during the nesting.

However, it must be noted that checking this option does not mean that nesting will use the mirror counterpart of the single-part. Nesting's main objective is to nest as many parts as possible and in so doing will try both the single part and its mirror counterpart to achieve that objective.

"Allow Part Mirror" can be used in conjunction with any of the above 4 options

Nest Half Quantity with Mirrored Parts

If this option is marked, the nesting will nest half the defined quantity in the specified orientation and the other half in mirrored image of the first.

If the quantity specified is "Maximize", the nesting will half the stock in the specified orientation and the other half the mirrored image. If there is remaining space, again the same logic applies – half in 1 orientation and the other a mirrored image.

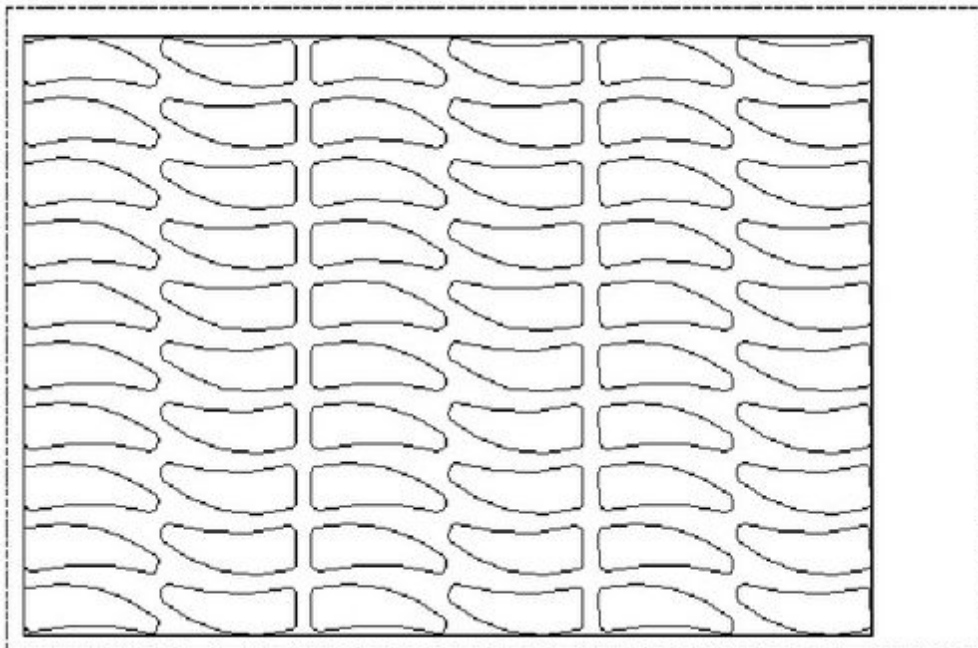
"Allow Part Mirror" has no bearing on this option, by default half of the total parts nested will be the mirror image.

TaskEdit - Users can enter multiple tasks but  each task at a time. To exit, click  button. For every task created, there are two corresponding files with extension **.JOB** & **.XML** being written on the default Task directory set in the **Sysdata**. For example:

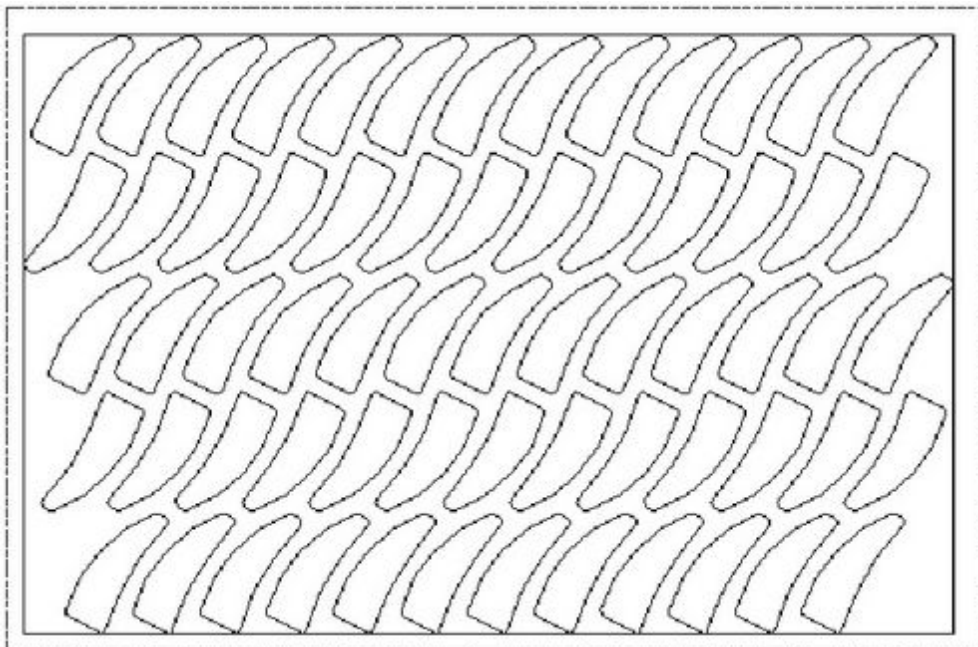
```
Task Name      :    SAMPLE
Filenames      :    SAMPLE.JOB
                :    SAMPLE_JOB.XML
```

Illustrations for Single Part Nest Control

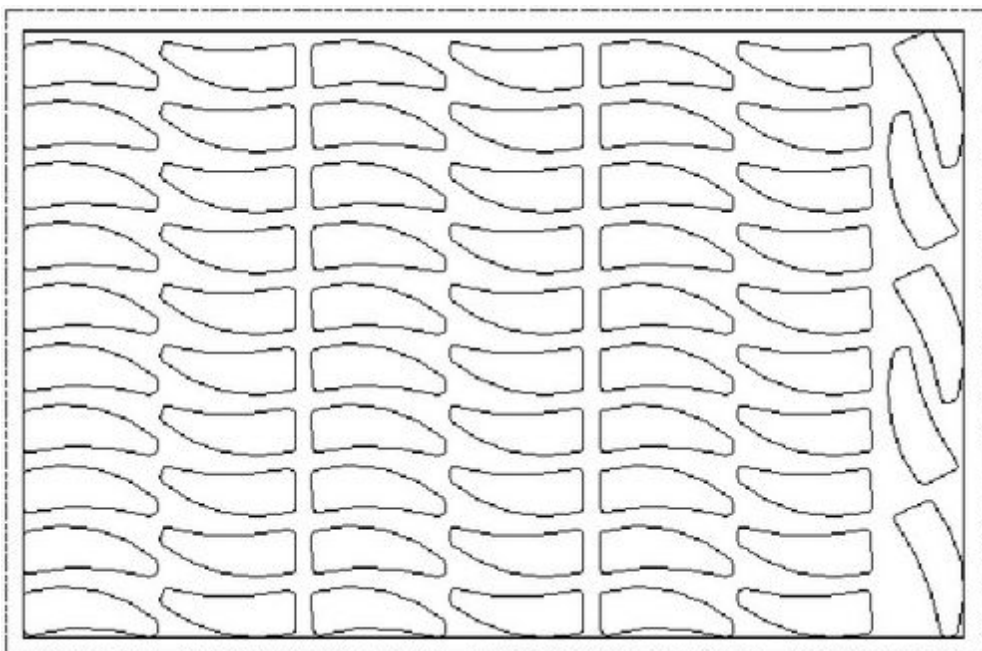
Density - Single Array



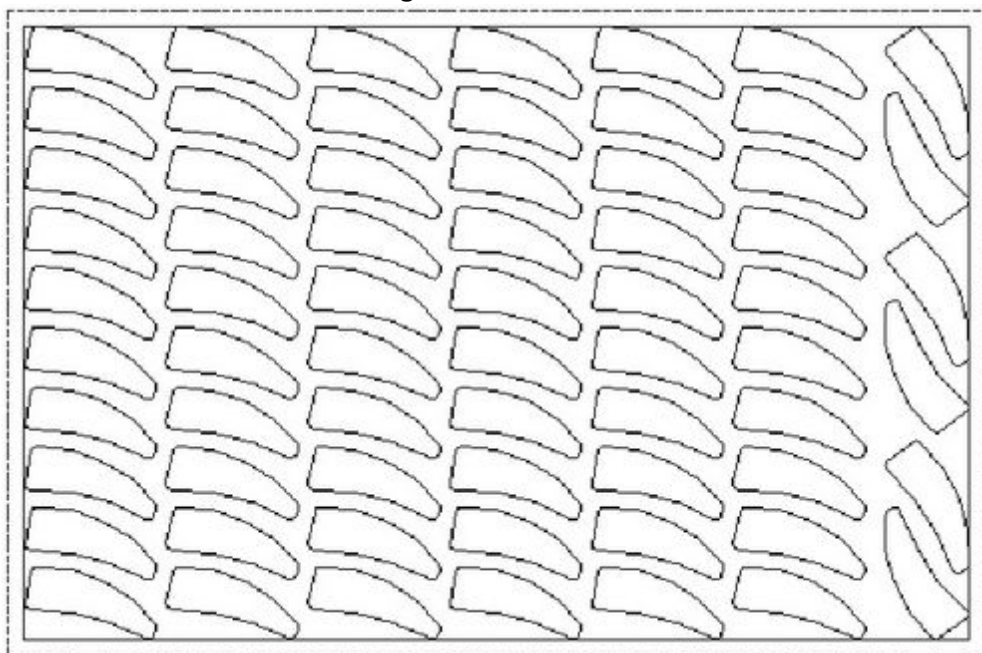
Quantity - Single Array



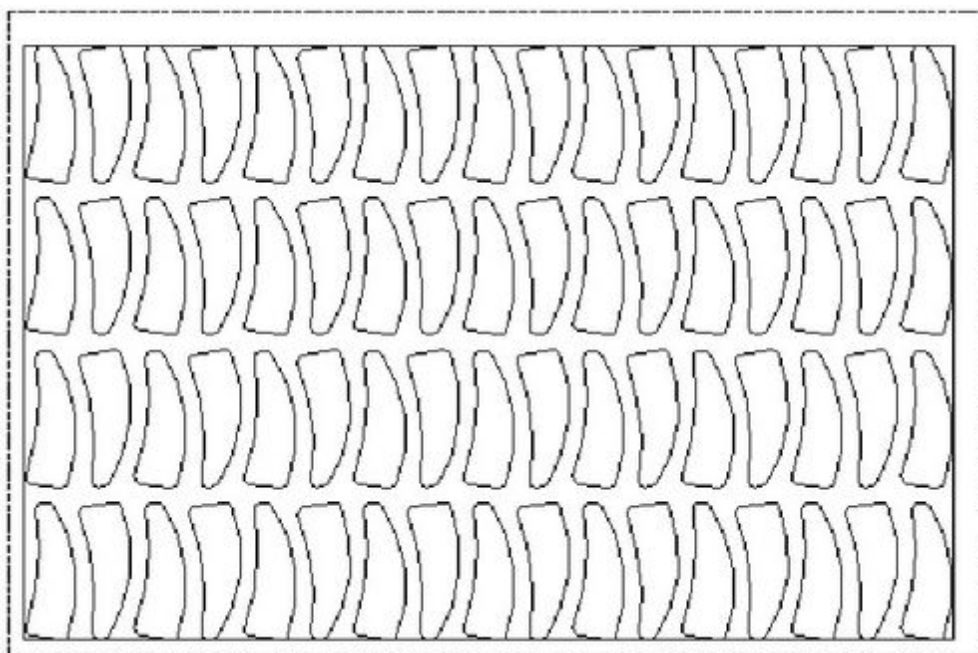
Quantity - Mixed Array



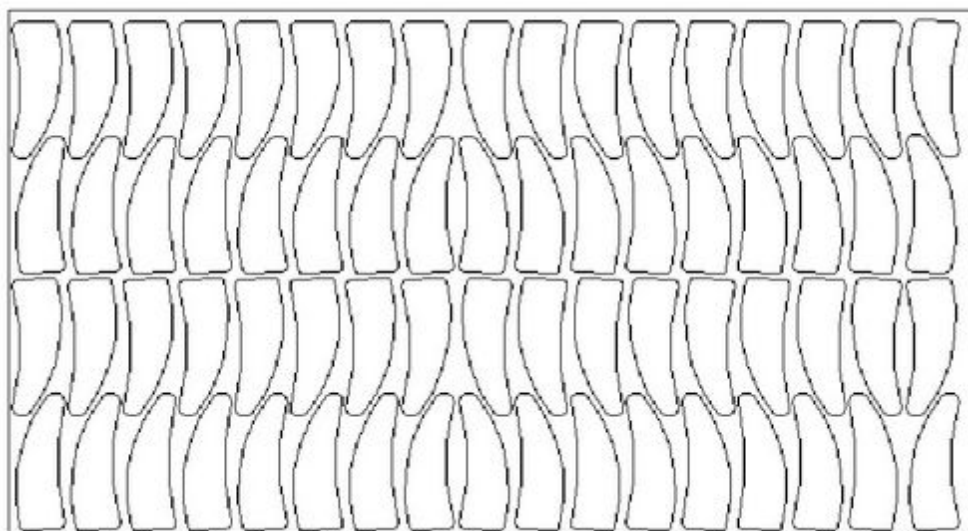
Extension- Long Nest



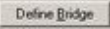
Allow Part Mirror (coupled with the **Quantity-Mixed Array** option)

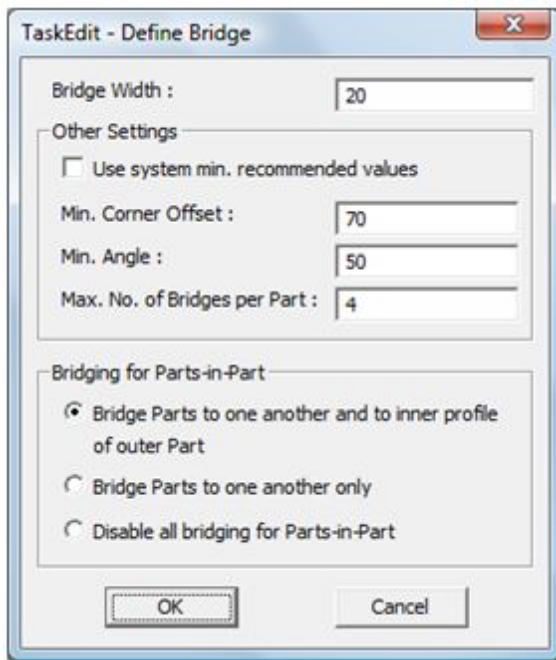


Nest Half Quantity with Mirrored Parts



Define Bridge 'E'

By clicking the  button, the dialog box below appears:



Bridge Width

A “BRIDGE” is the narrow strip that links one Part to another in order to maintain a continuous cutting process.

The width of the strip. Any numerical value is accepted, even zero.

Other Settings

These are the other bridge specifications that you can define for the nested tasks.

Use system min. recommended values Mark this check-box if you want to use the minimum system default values which are :-

Min. Corner Offset : 0.0

Min. Angle : 45

Max. No. of Bridges per Part : 5

Min. Corner Offset The minimum distance to maintain from the edge of Bridge to the nearest corner.

Min. Angle The angle formed by the Bridge and the part. The accepted range is 45 – 90 (degrees). See the following illustration.

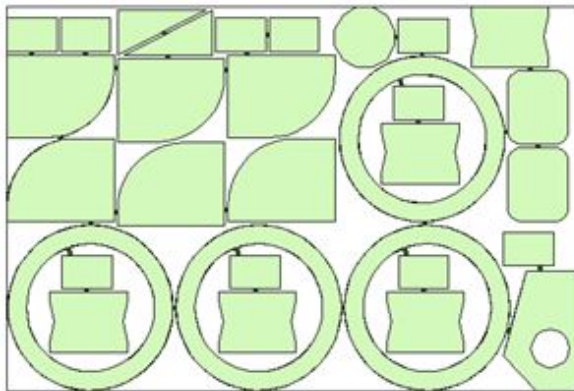
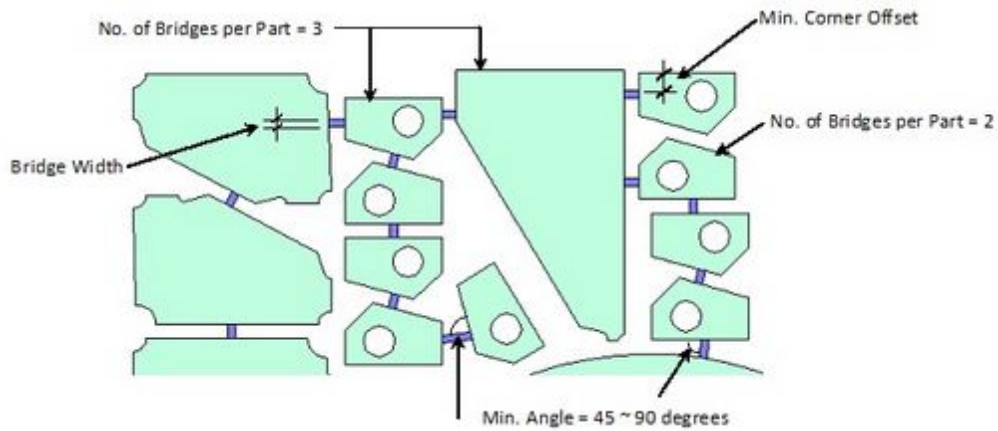
Max. No. of Bridges per Part The maximum number of bridges to be allowed to link to any one part. See the following illustration.

Bridging for Parts-in- Part (applicable for parts that are nested inside another Part)

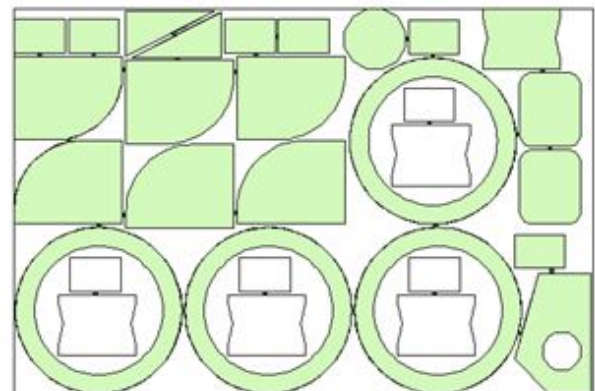
Bridge Parts to one another and to inner profile of outer Part This is the default setting. See illustration below – Option 1.

Bridge Parts to one another only See illustration below – Option 2.

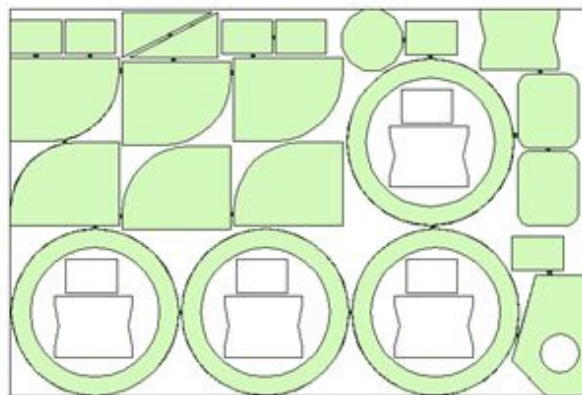
Disable all bridging for Parts-in-Part See illustration below – Option 3.



Option 1. Bridge Parts to one another and to inner profile of outer Part




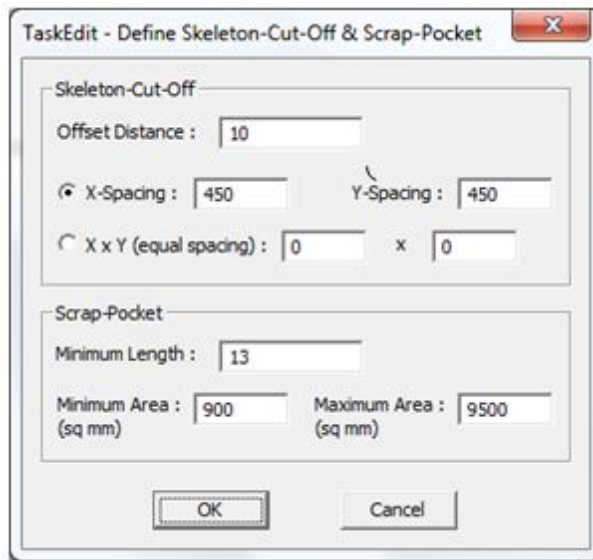
Option 2. Bridge Parts to one another only



Option 3. Disable all bridging for Parts-in-Part

Define Skeleton Cut-Off and Scrap Pocket 'F'

By clicking the button  the dialog box below appears:



Offset Distance

The gap between the skeleton-cut-off and the part profile (also called 'skeleton-bridge').

X-Spacing, Y-Spacing

X and Y spacings of the skeleton-cut-off.

X x Y (equal spacing)

Instead of entering the real values of the X and Y spacings, you can enter eg. **4 x 2** meaning the stock will be drawn such that there are 4 equal spacings in the X-direction and 2 equal spacings in the Y-direction.

Scrap-Pocket (see below for illustration)

Minimum Length

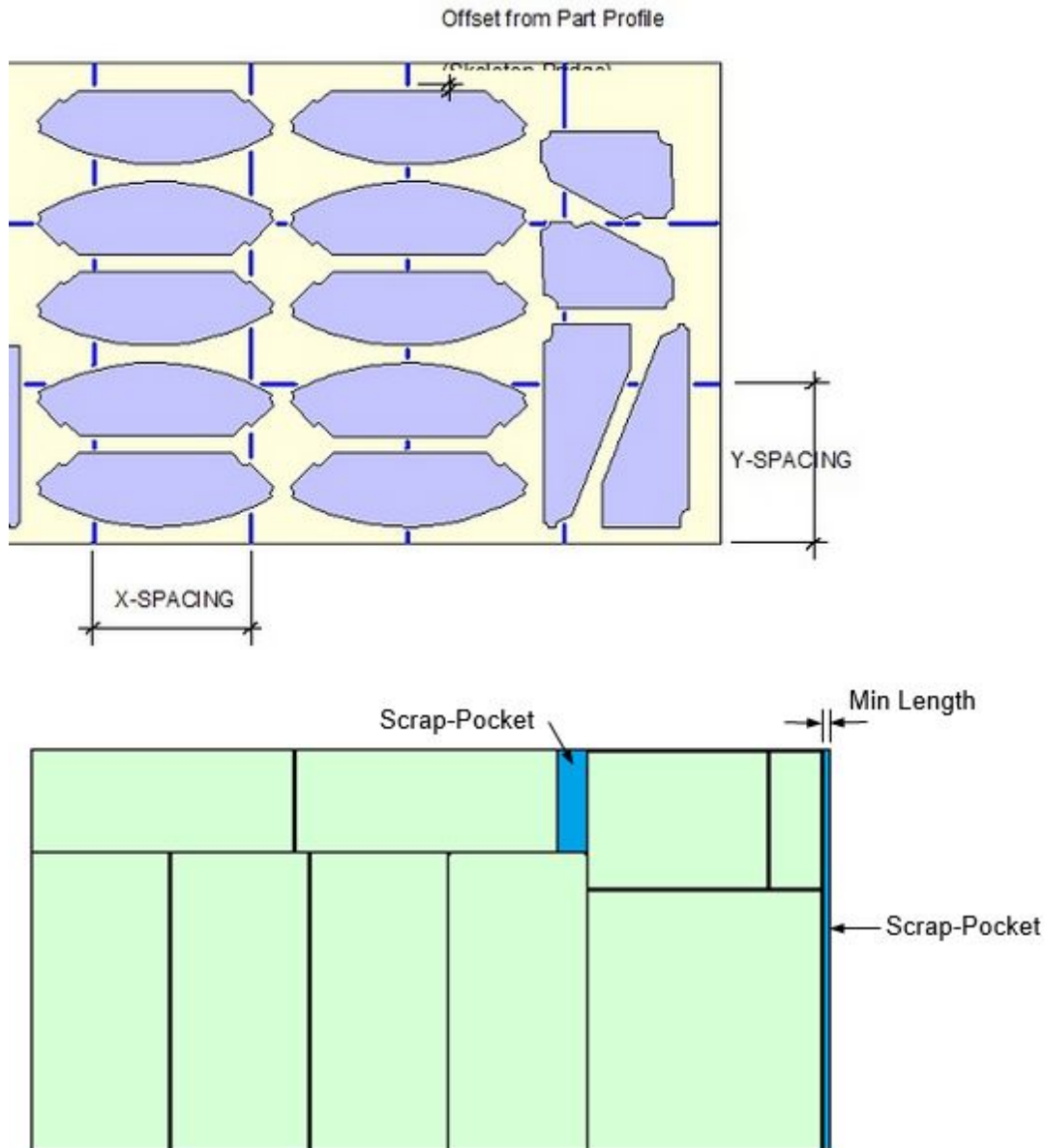
The length of any side of a Scrap-Pocket must not be smaller than this specified length.

Minimum Area

The area of any Scrap-Pocket must be bigger or equal to this specified area.

Maximum Area

The area of any Scrap-Pocket must not be bigger than this specified area.



Nest Now

This button will enable you to do nesting whilst in **TaskEdit**, without having to go to the **Nesting** or **BatchNesting** commands.

If one or both check-boxes-'**Auto-Bridge for Nested Layout**' and '**Skeleton-Cut-Off**' are marked, the layout(s) displayed will show the nested results with bridging/skeleton-cut-off.

View Bridge/Cut-Off

This button will enable you to view the nested layouts with Auto-bridge/Skeleton-Cut-Off.

Important : make sure that you have nested the Task first, before clicking this button.

This button is helpful if the Bridge/ Skeleton-Cut-Off specifications has been changed and you need to re-do the Bridging / Skeleton-Cut-Off– without having to go through the nesting process.

Unless one of the checkboxes for ‘Auto-Bridge for Nested Layout’, ‘Skeleton-Cut-Off’ or “Scrap-Pocket” is marked, this button will be greyed-out.

AutoNest Nesting

The core nesting engine of *AutoNEST* is called **NestPRO**. The basic technology of **NestPRO** lies in its powerful nesting engine. **NestPRO** features an intelligence processor that auto-adapts to different nesting conditions such as stocks and parts of different size and quantity. **NestPRO** generates nested layouts that reduces material wastage and maximizes utilization.

From version 9.9 or higher, there are two additional nesting engines. They are called “**Minimum Layouts**” and “**Minimum Layouts (Utilization)**”. These new engines are designed to handle nesting jobs with high quantity.

“**Minimum Layouts**” nesting engine was added to *AutoNEST* to cater for parts with high quantity. The objective is to achieve a nesting result with minimum nos. of layouts or high repeatability where materials are relatively inexpensive. See page 71 for illustration

“**Minimum Layouts (Utilization)**” is the latest nesting engine added to *AutoNEST* to improve the material utilization of the nesting results of “Minimum Layouts”. It has dual objectives – to achieve high repeatability of the layouts (thus reducing the no. of different layouts) and to optimize the material usage (reducing no. of sheets used). See page 71 for illustration.

From version 12.2 or higher, a fourth nesting engine is added. It is called “**Small-Part Control Nesting**” or **SPCN** for short. SPCN is designed for cutting machines with un-uniform suction where certain area(s) are weak in suction and therefore not suited for small parts to be nested on those area(s). Before this nesting engine can be run, “Small-Part Zone(s)” has to be defined on the stock size and the parts identified as “Small-Part” in the Task.

If you are unsure which engine to use – stick to the default which is **NestPRO**. **NestPRO** is able to nest most jobs efficiently with good results.

There are two commands for **Nesting**. Both commands generate nested layouts from tasks created by **TaskEdit**. The first is for nesting one task at a time, and the second, for batch nesting of multiple tasks.



Nesting (Single Task)

To nest a single task, one at a time and then displays the nested result onto the current AutoCAD drawing session.

**BatchNesting**

To nest a selection of tasks sequentially. Nested results in .DXF format are created for each nested task for viewing at a later stage.

If you are running in the AutoCAD environment, use the **ViewNEST** command to view the nested layouts.

AutoNest Nesting (Single Task)



Select the **Nesting** command from the icon or pulldown menu. The following will appear at the AutoCAD command prompt:

Insertion Pt <RETURN for (0,0)>:

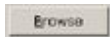
Select a point (lower-left corner) where the nested layout will be placed.

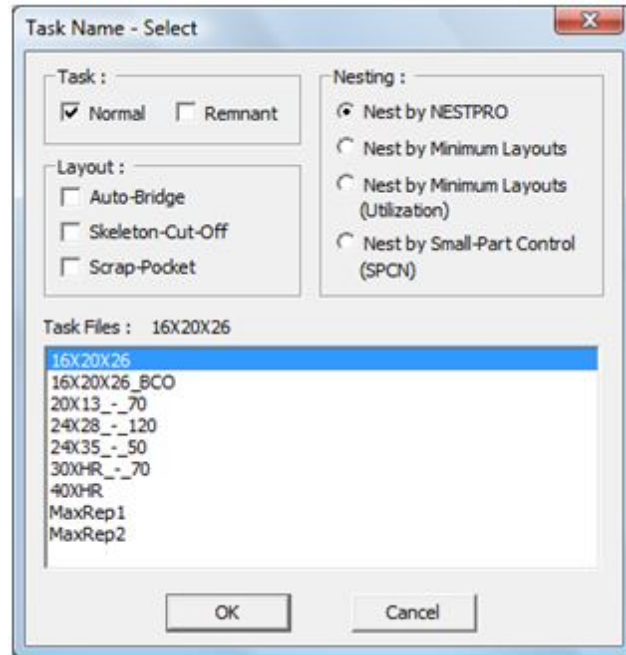
Subsequently, a dialog box as follows will appear.



Task Directory Display the default task directory set at **Sysdata**.

Task Name

Enter the task name to be nested or click the  button to display a list of tasks available in the task directory for selection as follows:



Task

Referring to the pop-up window above, there are 2 selections :

- ☐ Normal
- ☐ Remnant

“Normal” tasks refer to all task files (*.job) created by the user.

“Remnant” tasks refer to tasks files (*.job) with the reserved suffix, “_\$_REM” .
For example : XYZ_\$_REM.job.

“Remnant” tasks are automatically created if the “**Save Remnant**” check-box in the **Nesting Options** of *TaskEdit* is marked.

For more details, refer to [TaskEdit](#).

Nesting

There are 4 nesting engines for selection.

- **NESTPRO**
- **Minimum Layouts**
- **Minimum Layouts (Utilization)**
- **Small-Part Control (SPCN)**

The default is **NESTPRO** which is best suited for most jobs.

However when the parts'quantities are high, it is recommended you use “**Minimum Layouts (Utilization)**” nesting engine. This engine is designed to optimize the utilization of the stocks as well as to achieve a high repeatability of the layouts, thus minimizing the number of different nested layouts.

“**Minimum Layouts**” engine is only suitable when the stock materials are inexpensive and the objective is the maximize the repeatability of the nested layouts or cutting plan.

“**SPCN**” is only suitable for cutting machines with un-uniform suction where certain areas have weak suction. Therefore small-parts should not be nested over such weak suction area(s). This new nesting engine will ensure small-parts are nested within the designated “Small-part Zone”.

Layout

There are two check-boxes :

- ☐ Auto-Bridge
- ☐ Skeleton-Cut-Off
- ☐ Scrap Pocket

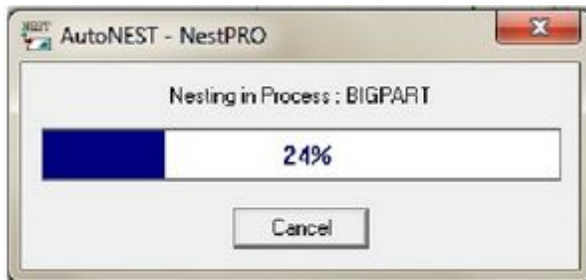
Mark the ‘Auto-Bridge’ check-box if you wish to include bridging into the nested layouts.

Similarly mark the ‘Skeleton-Cut-Off’ or “Scrap-Pocket” checkbox if you wish to include skeleton-cut-off’s or Scrap-Pockets into the nested layouts.

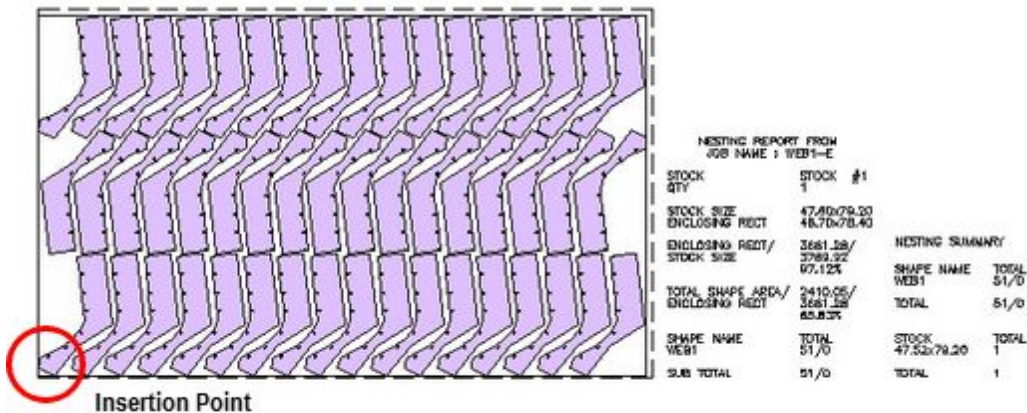
AutoBridge, Skeleton-Cut-Off and Scrap-Pocket will follow the specifications as defined in **TaskEdit**.

For more details, refer to [TaskEdit](#).

Once the Task name has been confirmed, click the **OK** button to start the nesting. The following dialog box appears – indicating that the nesting is in progress.



The nested layout will be displayed onto the current drawing, at the location indicated by the Insertion point.



Once a task has been successfully nested, output files will be created on the same folder as the Task. These files have the same name as the Task but different file extensions as shown below.

Example : Task name is *SAMPLE*

Output Files:

SAMPLE.SYM
SAMPLE.SUM
SAMPLE.XLS or XLSX
SAMPLE_SYM.XML
SAMPLE_SUM.XML

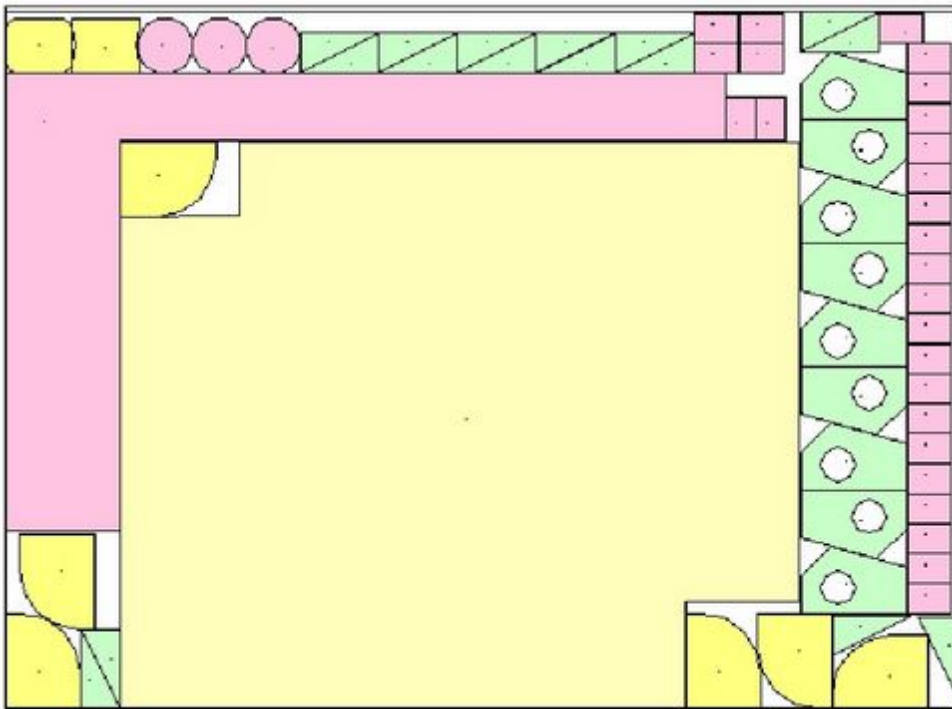
The ***.SYM*** file contains the results of the nesting in terms of each part and its location in the layout. The formats of above files are described in detail in [SYM File Format](#) section.

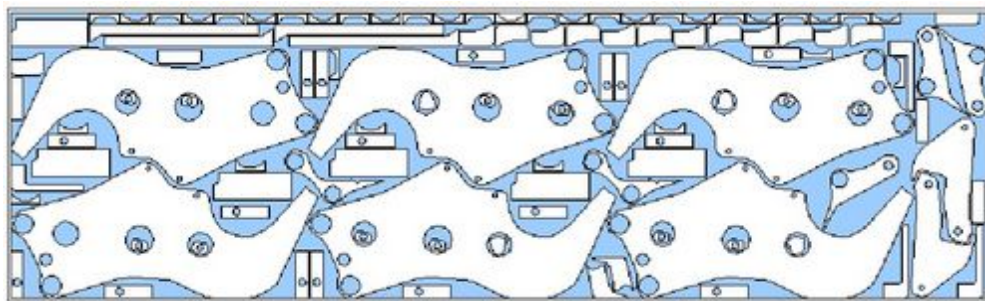
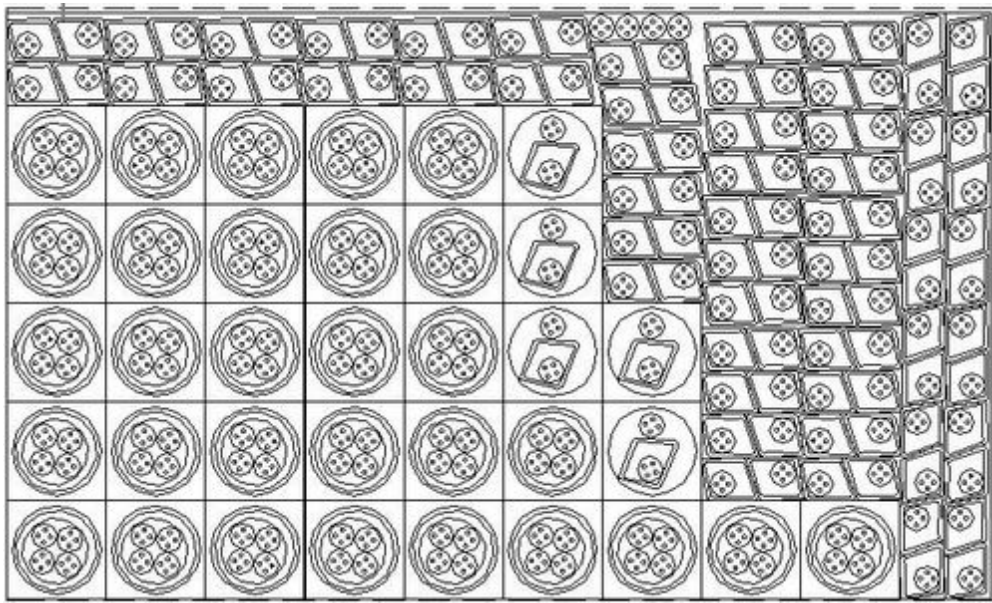
The ***.SUM*** file contains the nesting summary report in TEXT format.

The ***.XLS or .XLSX*** file is an Excel spreadsheet of the nesting summary.

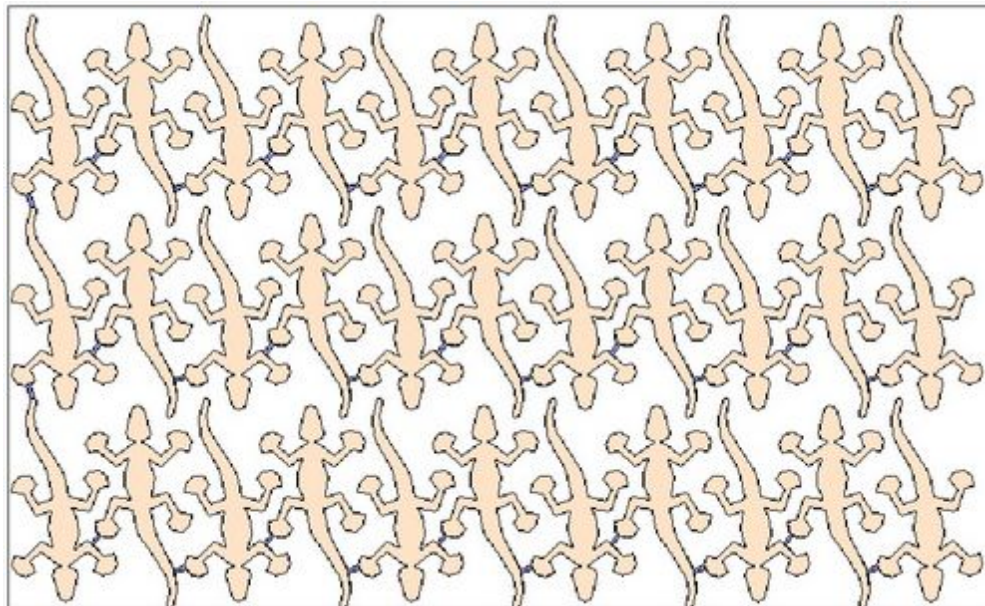
The two ***.XML*** files contain the same information as ***.SYM*** and ***.SUM*** but written in extensible markup language (XML).

Examples of Nested Layouts:

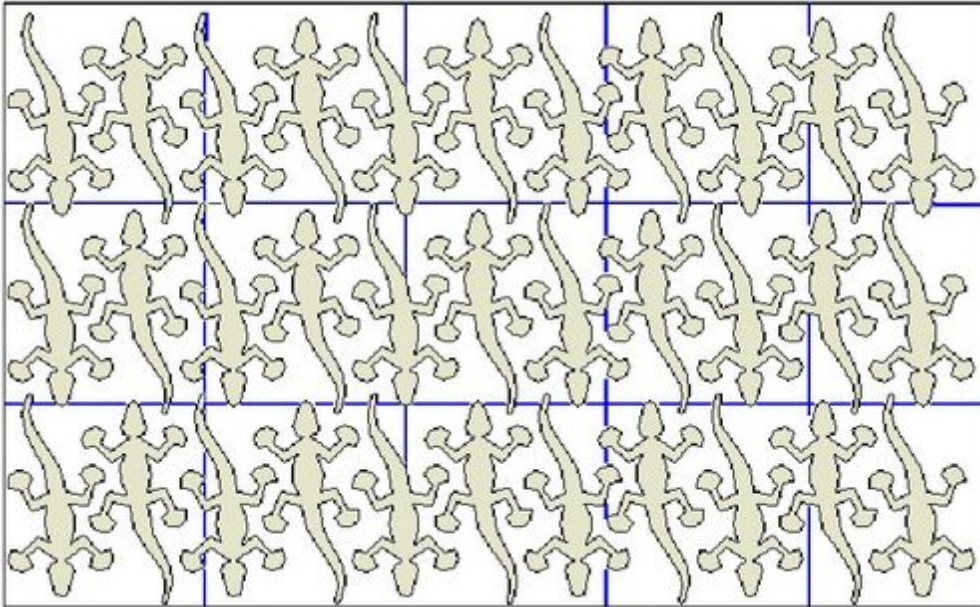




Nesting Layout with Auto-Bridge (Staydown):



Nested Layout with Skeleton-Cut-Off:



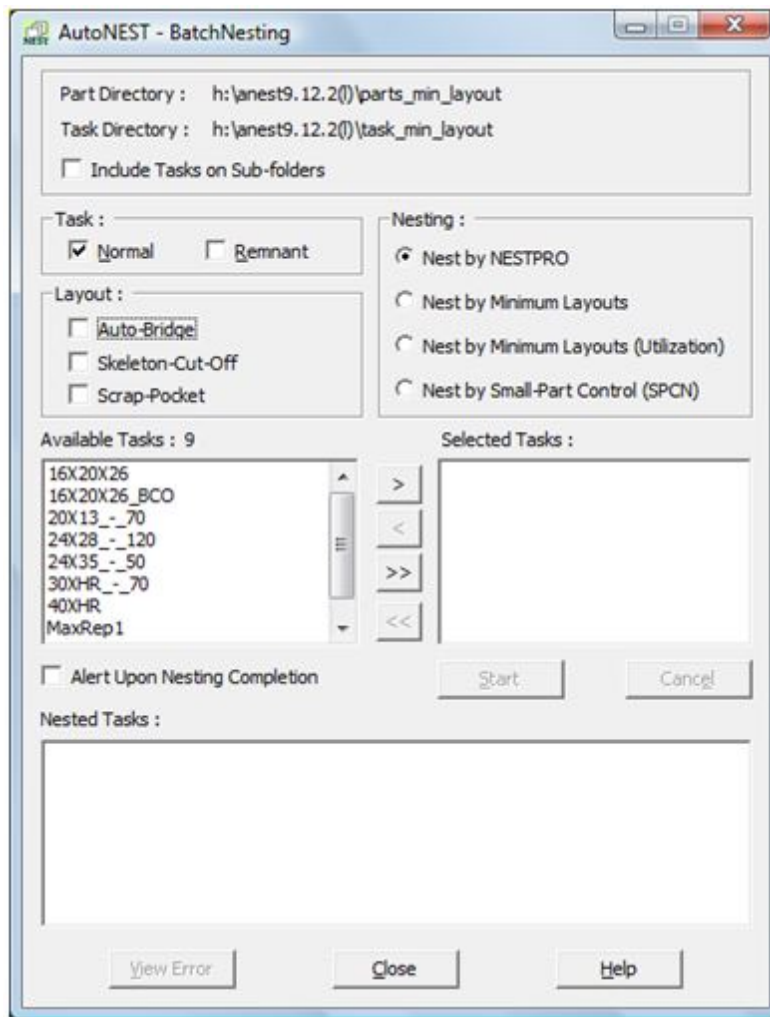
AutoNest Batch Nesting



At times you may like to nest a series of tasks overnight or during a break. This facility is provided via the **BatchNesting** command. But to view the already nested layouts, select the **ViewNEST** command if you are working in AutoCAD. In **Nest Manager**, just open each of the nested results saved in .DXF files from the 'Task Directory' using a CAD or a DXF Viewer.

If the "Include Sub-Folders" check-box is marked, **BatchNesting** is able to find and nest Tasks (.JOB) residing on sub-folders.

Select the **BatchNesting** command from the icon or pulldown menu. The following dialog box will appear:



Each dialog box input option is described in the following:

Parts Directory Display the default part directory as set in the *Sysdata*.

Task Directory Display the default task directory as set in the *Sysdata*.

Include Tasks on Sub-Folders

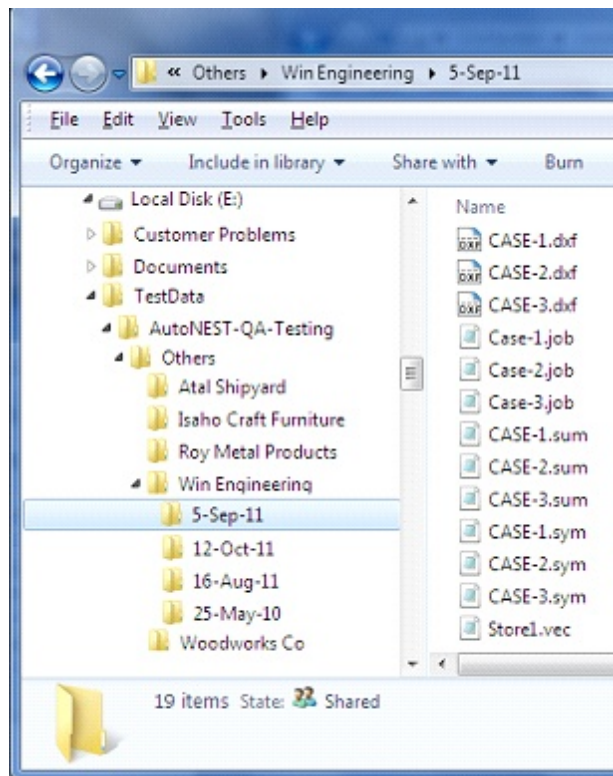
When this check-box is marked, all the task files residing on all the subfolders of the specified Task Directory will be displayed onto the “Available Tasks” list box.

For example :

Parts Directory : E:\TestData\AutoNEST-QA-Testing\Others

Task Directory : E:\TestData\AutoNEST-QA-Testing\Others

All task files (.JOB) residing on subfolders, to a maximum of two levels from ...\\Others will be displayed for selection.



Task

There are 2 check-boxes:-

- ☐ Normal
- ☐ Remnant

‘**Normal**’ tasks refer to all the task files (*.job) created by the user.

‘**Remnant**’ tasks refer to tasks files (*.job) with the reserved suffix, ‘_\$REM’. For example : [sample_\\$REM.job](#). These tasks are automatically created if the “**Save Remnant**” check-box in the **Nesting Options** of **TaskEdit** is marked.

For more details, refer to [TaskEdit](#).

Nesting

From version 9.12.2 or higher, there are 4 nesting engines :-

- Nest by NESTPRO
- Nest by Minimum Layouts
- Nest by Minimum Layouts (Utilization)
- Nest by Small-Part Control (SPCN)

You can select the nesting engine of your choice. Detailed explanation of each engine is explained in Chapter 5.5 Nesting.

The default is **NESTPRO** which is best suited for most jobs.

However when the parts' quantities are high, it is recommended you use "**Minimum Layouts (Utilization)**" nesting engine. This engine is designed to optimize the utilization of the stocks as well as to achieve a high repeatability of the layouts, thus minimizing the number of different nested layouts.

"**Minimum Layouts**" engine is only suitable when the stock materials are inexpensive and the objective is the maximize the repeatability of the nested layouts or cutting plan.

"**SPCN**" is only suitable for cutting machines with un-uniform suction where certain areas have weak suction. Therefore small-parts should not be nested over such weak suction area(s). This new nesting engine will ensure small-parts are nested within the designated "Small-part Zone".

Layout

There are three checkboxes :-



- ☐ Auto-Bridge
- ☐ Skeleton-Cut-Off
- ☐ Scrap-Pocket

Mark the 'Auto-Bridge' checkbox if you wish to include bridging into the nested layouts.



Similarly mark the 'Skeleton-Cut-Off' or "Scrap-Pocket" checkbox if you wish to include skeleton-cut-off's or scrap-pockets into the nested layouts.

AutoBridge, Skeleton-Cut-Off and Scrap-Pocket will follow the specifications as defined in [TaskEdit](#).

Available Tasks

Display a list of tasks (**.JOB**) files found in the task directory. You can select one or more files to be nested by highlighting the file names and then click the  button. Click the  button to select **ALL** the files.

Selected Tasks


Display a list of task files selected to be nested. Click the  button to un-select the file(s) or click  to un-select ALL the files in the list box.

You can click the  button to start the nesting process.

Alert Upon Nesting Completion

When this check-box is marked and when the batch nesting process is completed, **BatchNesting** will play the Windows default sound three times to alert the user (provided the speakers are correctly connected to the computer and turned-on).

Nested Tasks

Display a list of successfully nested task files. Click the  button to see if there is any error message.

Internally, once a task has been successfully nested, output files will be created. These files have the same name as the task but different file extensions as shown below.

Example : Task name is *SAMPLE*

Output Files: *SAMPLE.SYM*
SAMPLE.SUM

SAMPLE.XLS or .XLSX
SAMPLE_SYM.XML
SAMPLE_SUM.XML

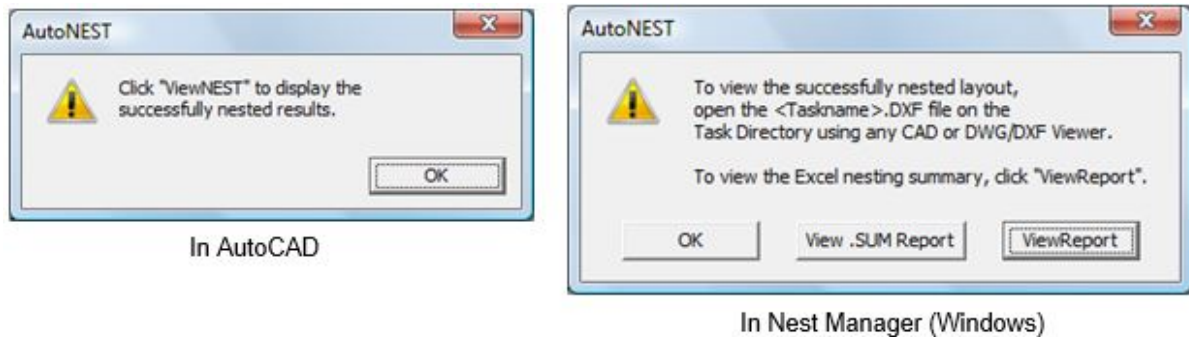
The **.SYM** file contains the results of the nesting in terms of each part and its location in the layout.

The **.SUM** file contains the nesting summary report in TEXT format.

The **.XLS or .XLSX** file contains the nesting summary in Excel format.

The two **.XML** files contain the same information as .SYM and .SUM but written in extensible markup language (XML).

Upon completion of the nesting, the dialog box below will be displayed:



In **AutoCAD**, when the nesting has completed, you may use the **ViewNEST** command to view the nested results or open each of the nested layout (DXF files) from the Task directory.

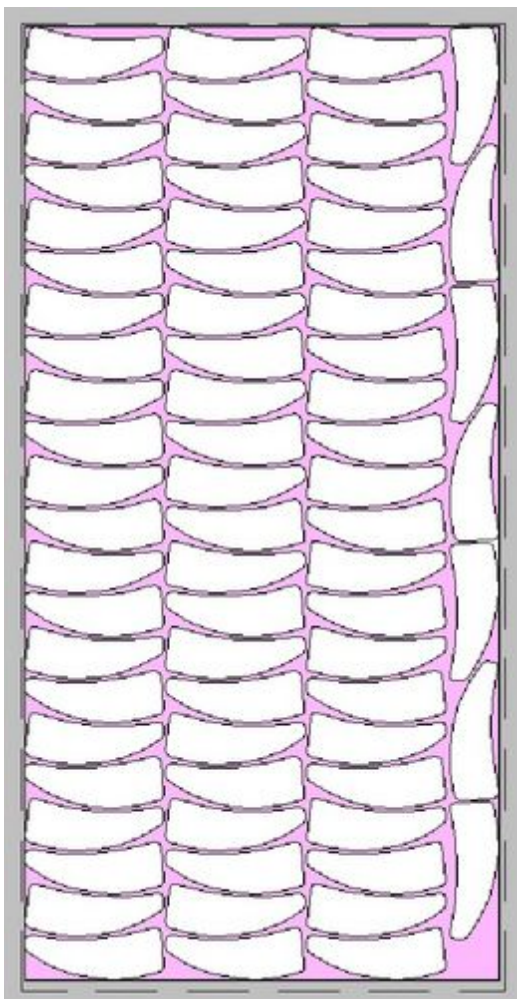
In **Nest Manager**, upon the completion of the nesting, you can open the nested layout (DXF file) from the Task directory using a CAD or a DXF Viewer. To view the report of each of the nested results, click the **“View .SUM Report”** button or the **“View Report”** button to view the Excel report.

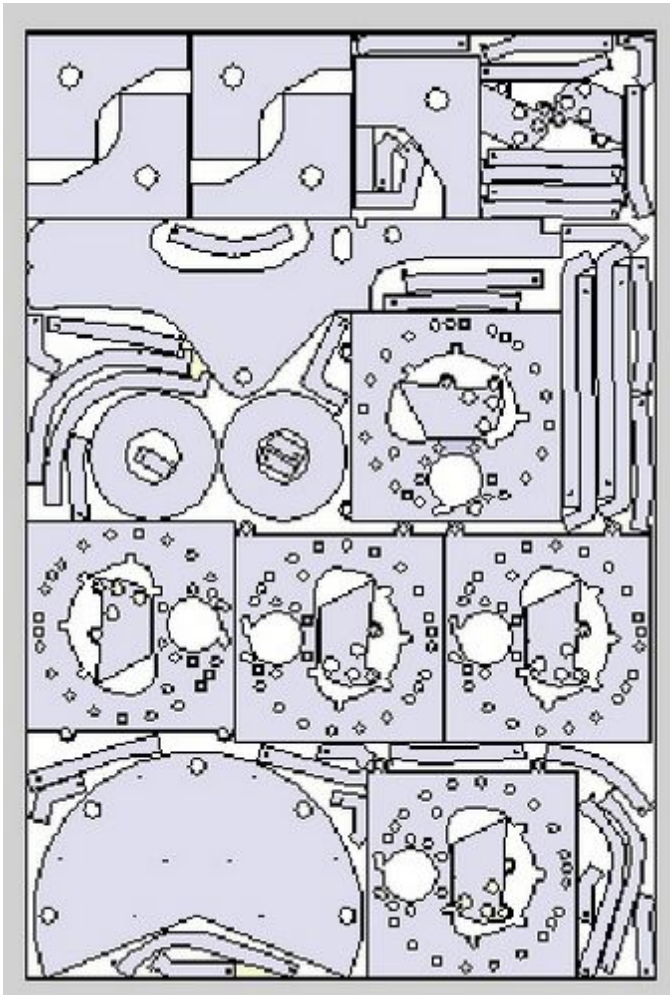
Notes on Nested Layouts

1. The display of the nested results will be determined by the settings in “Nested Layout – Presentation” and “Nested Layout – Layer / Color Setting” of **Sysdata**.
2. You can choose to display the parts in their original layers /colors by selecting “DXF” or “DWG” in “File Format” of **Sysdata**.
3. Conversely you can choose to display the parts in “VEC” File Format where you can determine the layers /colors of:
 - External or outer profile of the Part
 - Internal or inner profile of the Part

Go to [Sysdata](#) settings for more details.

Sample of Nesting Layouts:





AutoNest View Nest



The **ViewNEST** command is used to output nested task layout(s) (*.sym) onto the current AutoCAD session as a drawing. This command is especially useful for viewing of already nested layouts (e.g. **BatchNesting**), viewing of updated nested layouts when there is change in the report text size or output file format type (as specified in **Sysdata**) - without having to re-nest the task again.

ViewNEST will enable you view nested results with Auto-bridge/ Skeleton-Cut-Off.

ViewNEST will prompt :-

View Nesting Results:

Insertion Pt <RETURN for (0,0)>:

Select a point (lower-left corner) where the nested layout will be placed.

Nested File Name (? for list) <> :

Enter the nested Task name or enter "?" to select from a window pop-up, showing the nested files from the TASK directory.

Nested Layout to Include [Auto-Bridge/None] <None> :

Enter "A" to include bridging with the nested layouts, otherwise just press the ENTER key.

Include [Skeleton Cut-Off/SCrap-Pocket/Both/None] <None> :

Enter "S" to include Skeleton-cut-off, "SC" to include Scrap-pocket or "B" to include both Skeleton-cut-off and Scrap-pocket into the nested result. The default is None.

Proceed to view another Nesting result? (Y/N) <N> :

This prompt will allow you to view another nested result without quitting the command.

Nesting Summary Report

There are 2 reports :- "Layout Summary" and "Task Summary". These are created for each nested task at two places.

- a) Near to the nested layout (default : right-hand side of layout)
- b) Text file in a . **SUM** file

The position of these reports displayed on the drawing is customizable in ANEST.SET.

You can set the TEXTSIZE of these nesting reports on the drawing through the **Sysdata**. In the same command, you can also choose NOT to display it with the nested layout. As for the layer and color of the reports, you can define them in "Layer / Color Setting" of **Sysdata**.

The following information will be presented in the Nesting summary report:-

- i.) Name and quantity of parts nested / unnested.
- ii.) Total area and perimeter of parts nested
- iii.) $\frac{\text{Area of enclosing rectangle of nested parts}}{\text{Area of stock sheet}} \times 100 \%$
- iv.) $\frac{\text{Sum of area of all parts nested}}{\text{Area of enclosing rectangle of nested parts}} \times 100 \%$
- v.) Stock Utilization (area of parts / area of stock x100) %

Sample "Layout Summary" Report

NESTING REPORT OF TASK NAME : PIP

STOCK

STOCK #1

QTY 1
COST 99.00

STOCK SIZE 2423.20x6121.50
ENCLOSING RECT 2291.70x5966.50

ENCLOSING RECT 13672831.70 /
STOCK SIZE 14833373.94
92.18%

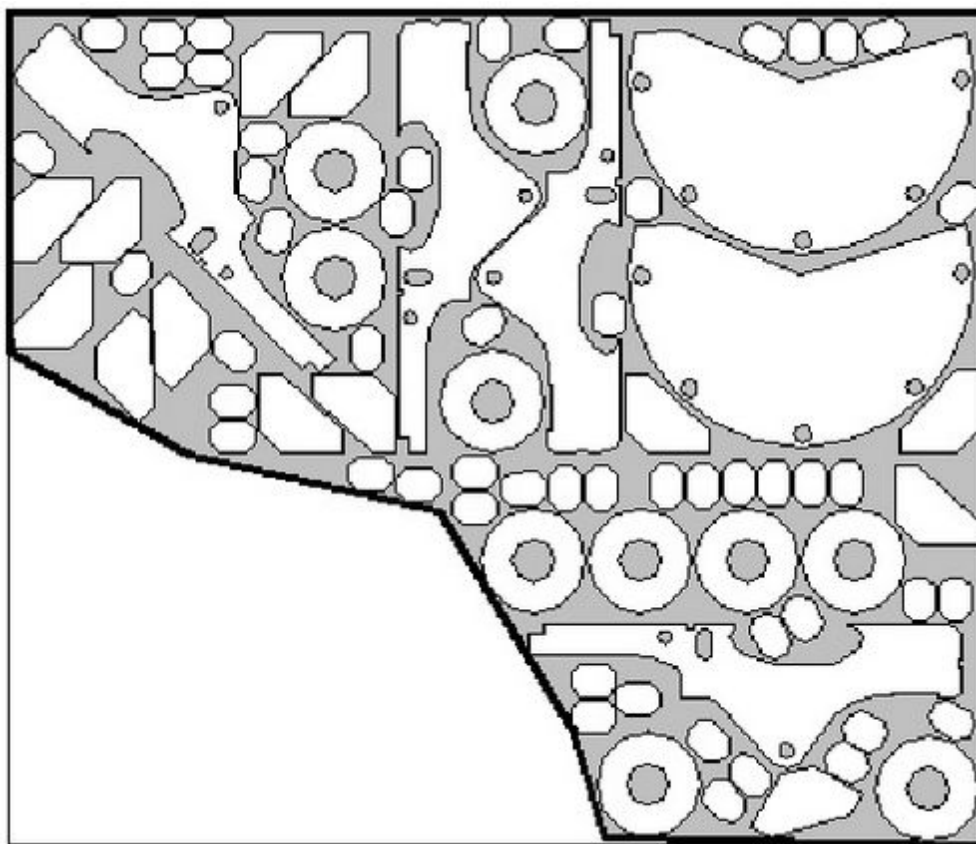
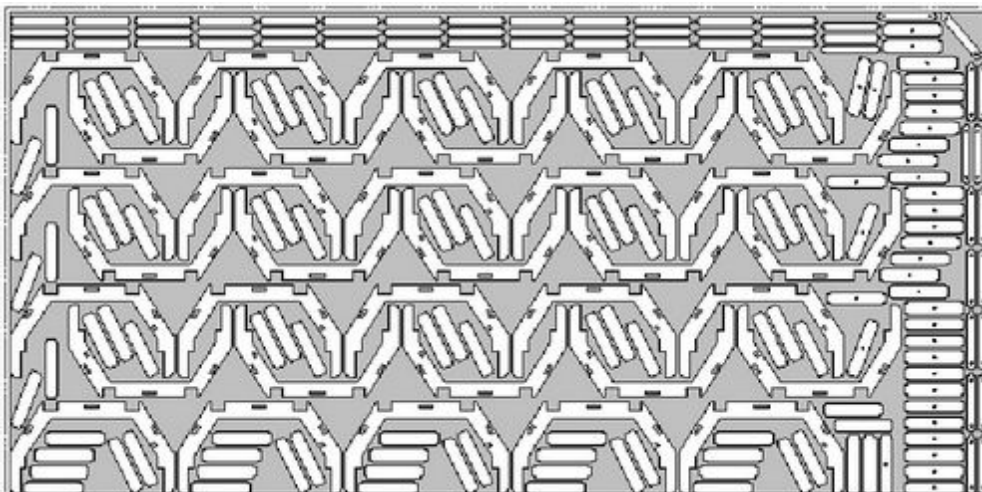
TOTAL PART AREA/ 9400020.32/ NESTING SUMMARY
ENCLOSING RECT 13672831.70

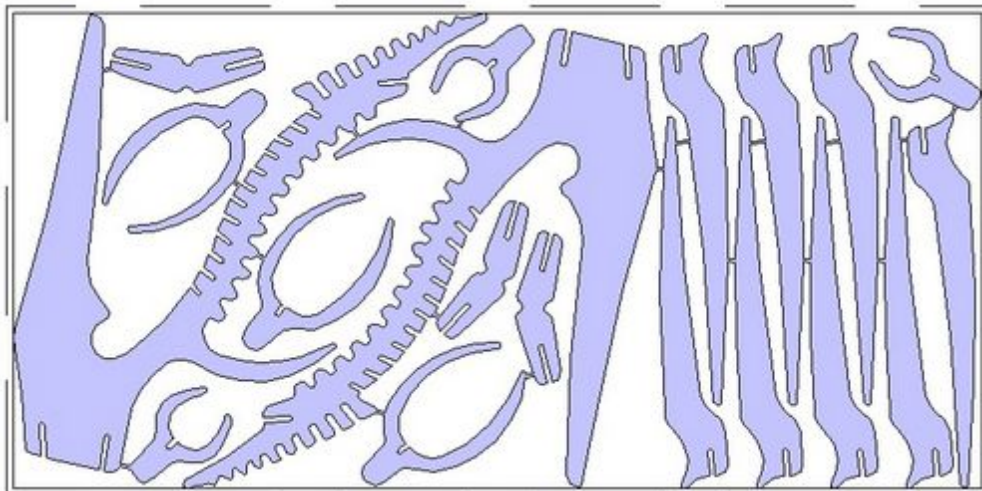
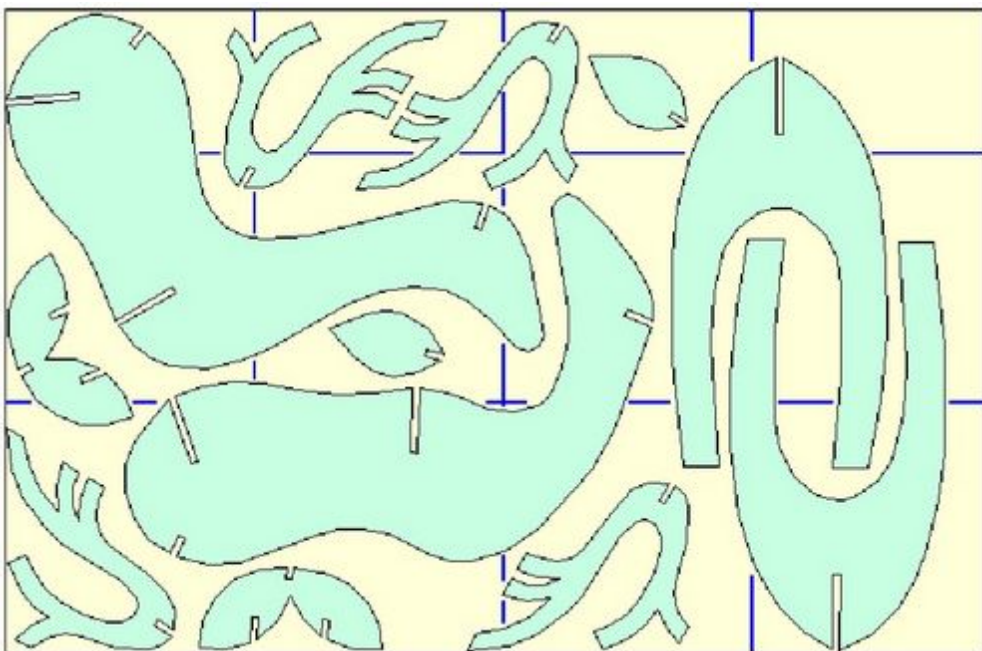
	68.75%	PART NAME	BASIC QTY	FILLER QTY
STOCK UTILIZATION	63.37%	PART1	6/6	-
		PART2	12/12	-
TOTAL PART		PART3	24/24	-
PERIMETER	195202.62	PART4	18/18	0/ 50
		PART5	18/18	-

PART NAME

PART1	6/6	TOTAL PARTS NESTED	78/ 78	0/ 50
PART2	12/12			
PART3	24/24			
PART4	18/18	STOCK	QTY	COST
PART5	18/18	2423.16x6121.50	1	99.00

SUB TOTAL	78/ 78	TOTAL	1	99.00
-----------	--------	-------	---	-------

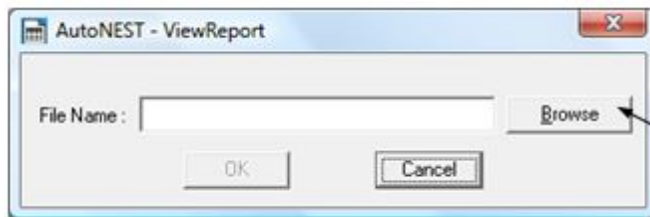
Irregular Stock Nested Layout:**Typical nested layouts:**

Nested Layout with Auto-Bridge (Staydown):**Nested Layout with Skeleton-Cut-Off:****Anest View Report**

This command will enable to you to read the Excel report of the nesting summary generated upon a successful nesting.

This command will only works if there is a Microsoft Excel installed on the computer that you are running AutoNEST.

If you are running AutoNEST in **Nest Manager**, select the **ViewReport** command icon and the dialog below will appear.

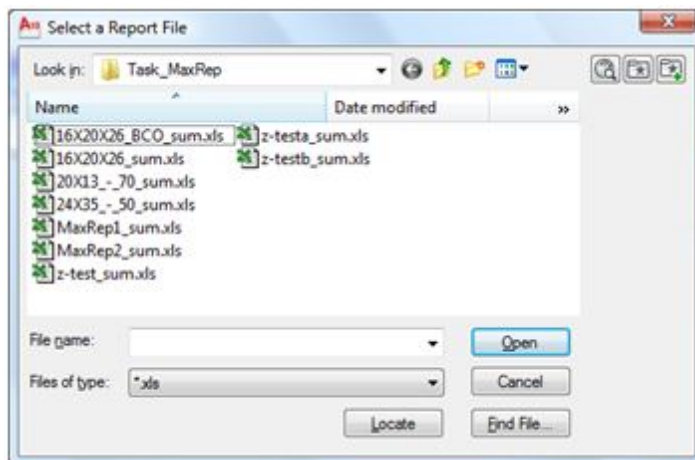


Click the 'Browse' button and the following dialog box will appear.

If you are running AutoNEST on AutoCAD, select the **ViewReport** command icon from the toolbar or from the AutoNEST pulldown. The following prompt is displayed:

View Nesting Report:
Report File Name (? for list)<>:

Enter ? and the following dialog box will appear.



Depending on your Excel version, you are able select the Excel nested report summary 'Taskname.xls' or 'Taskname.xlsx' to view.

The first screenshot shows the 'Layout#1' report for task 20X13_70. It lists dimensions for stock, stock size, and enclosing rectangle, along with calculated areas and perimeters.

The second screenshot shows the 'Layout#2' report for task 20X13_70, providing similar data for a second layout.

The third screenshot shows the 'Summary' report, which consolidates data from both layouts into a single table.

1	NESTING REPORT OF TASK NAME : 20X13_70				
2					
3	STOCK	STOCK #1	STOCK #2		
4	QTY	2	2		
5					
6	STOCK SIZE	2370 00x1970 00	2370 00x1970 00		
7	ENCLOSING RECT	1928 32x2330 00	1780 00x1350 00		
8					
9	ENCLOSING RECT	4492997 05	2403000		
10	STOCK AREA	4668900	4668900		
11	ENCL RECT/ STK AREA	96 23%	51 47%		
12					
13	TOTAL PART AREA	4149028 79	1937250		
14	ENCLOSING RECT	4492997 05	2403000		
15	TOTAL PART AREA/ ENCL RECT	92 34%	80 62%		
16					
17	TOTAL PART AREA/ STK AREA	88 87%	41 49%		
18					
19	TOTAL PART PERIMETER	38815 9	33090		
20					
21	PART NAME				
22	20X13_0	4	0		
23	20X13_1	1	0		
24	20X13_2	5	3		
25	20X13_3	0	12		
26	20X13_4	6	0		
27	SUB TOTAL	16	15		
28					
29					
30					
31					
32					
33					
34					
35					
36					
37					
38					
39					
40					
41					
42					
43					
44					
45					
46					
47					
48					
49					
50					
51					
52					
53					
54					
55					
56					
57					
58					
59					
60					
61					
62					
63					
64					
65					
66					
67					
68					
69					
70					
71					
72					
73					
74					
75					
76					
77					
78					
79					
80					
81					
82					
83					
84					
85					
86					
87					
88					
89					
90					
91					
92					
93					
94					
95					
96					
97					
98					
99					
100					

The report of each nested layout will be listed on each worksheet. If there are 2 nested layouts, the Excel file will have 2 worksheets (named Layout#1 and Layout#2) and a 3rd worksheet named "Summary".

Relevant information such as area, perimeter and quantity of nested Parts; Stock quantity used, Stock balance; Utilization (area of parts/area of stock) ... are tabulated.

AutoNest Update Nest



Once the nested layout is presented onto the current AutoCAD drawing, you can have full access to all the AutoCAD commands. It is sometimes easier for the user to scrutinize an already nested layout and then decide how it can be further improved manually. In this case you can use your better judgement and experience to edit the layout through the **MOVE, COPY, ...etc.** commands of AutoCAD.

However, after such a session, *AutoNEST* is not aware of these changes. Therefore, an **UpdateNEST** is necessary so that *AutoNEST* will do the following:

- Update the drawings by re-displaying the nested layouts
- Update the nesting summary and re-calculate the utilization percentages
- Update the .SYM and .SUM files

NOTE: This command works only if the nested layout is displayed in DWG format (see Sysdata's "File Format")

The prompts below will appear :-

UpdateNEST: Task Name (? For list)<>:

The user is prompted the Task name that needs to be updated. < > will contain the name of the most recent entry of the Task name.

New Nested File Name<>:

You will be prompted the name of the new nested (.sym and .sum) file. You can enter a new name (e.g. SAMPLE-a) or if you enter the same name (e.g. SAMPLE), then the **UpdateNEST** will overwrite the existing nested files of the same name.

Select Objects:

Here you are requested to enclose the layout(s). If the task consists of several nested layouts and only one of them has been edited, you still have to enclose/select all of them as long as they are of the same task.

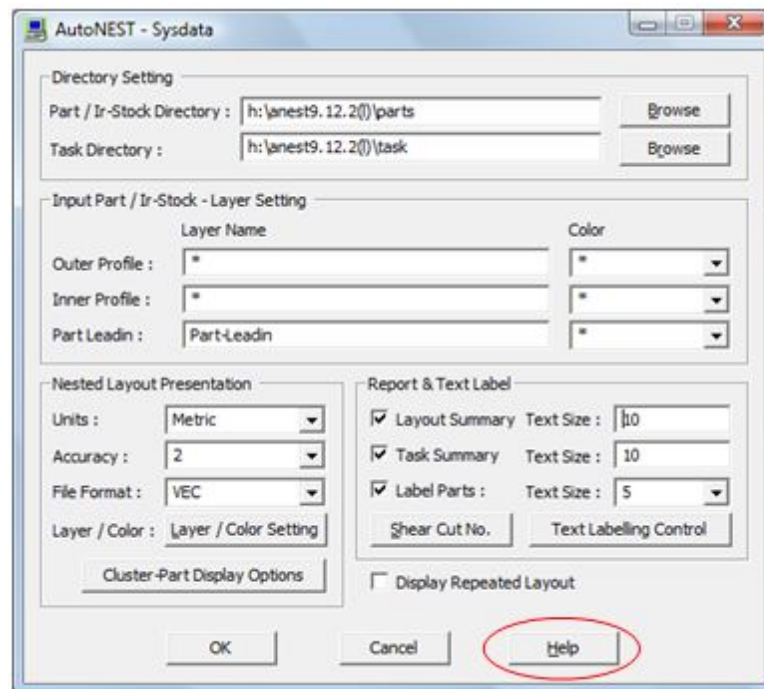
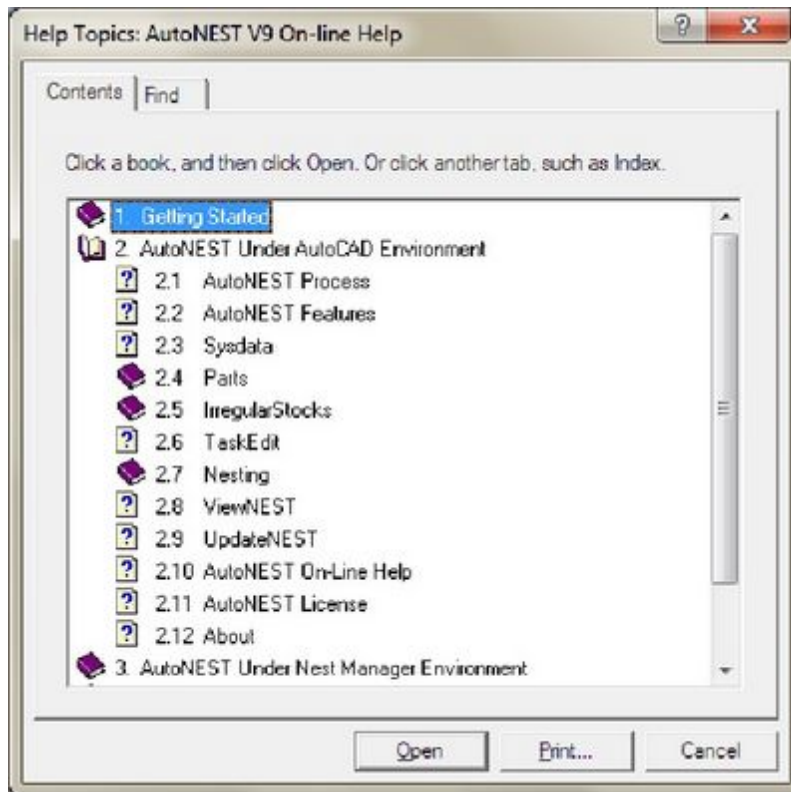
Insertion Pt of New layout <RETURN for (0,0)>:

Indicate a point on the screen to indicate the lower-left corner of the nested layouts. The updated nested layouts and nested summary reports will be displayed with reference to the new insertion point.

AutoNest On-Line Help



This command will display AutoNEST on-line documentation. In addition for each command, you can click the **"Help"** button which is usually located on the lower-right corner of the dialog box.

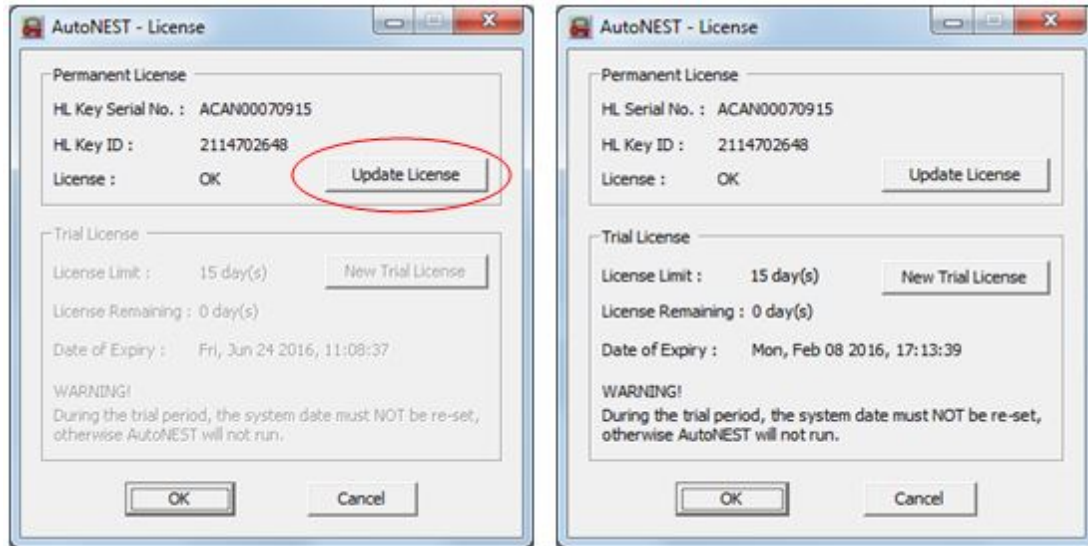


Help button on each of the dialog boxes.

AutoNest License



This command will display the current status of *AutoNEST* license. Upon a successful installation, there is a 15 day Free trial to run *AutoNEST* without any hardware protection device.



Standalone :

PERMANENT License

"HL Key Serial No" - Shows the hardware lock's serial no. There should not be any space in between the numbers.

"HL Key ID" - This is the unique ID of your usb HL key

"License" – You should see "OK" for it indicates that both the hardware lock and the authorization file are in working order.

"Update License" - This button is used to upgrade the existing AutoNEST license using a "RadanLic.UPD" file.

TRIAL License

"License Limit" - This indicates the maximum number of days available to run AutoNEST without a permanent license.

"License Remaining" - indicates the number of remaining days to run AutoNEST without permanent license.

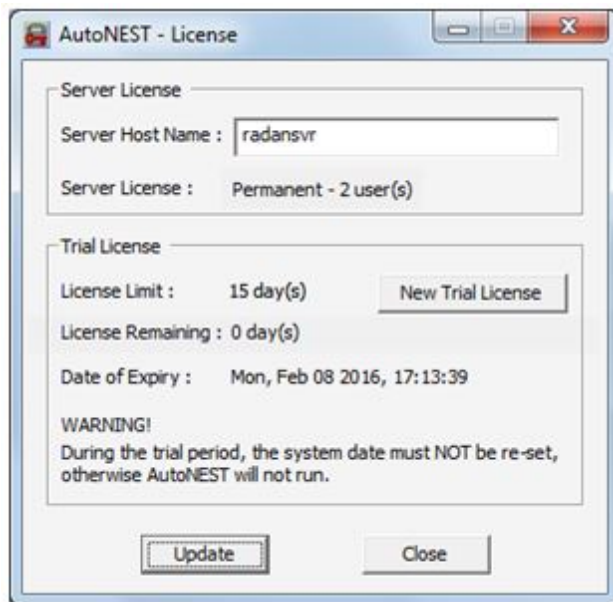
"Date of Expiry" - This shows the exact expiry date and time of the existing Trial License.

"New Trial License": - This button is used to extend the trial period using a ".V2C" file.

Network : Server License

On the Client

This command will display the current status of **AutoNEST license on the Client machine**. Upon a successful installation, there is a 15 day Free trial to run AutoNEST without any protection device.



"Server Host Name" - Host name of your Server (installed with AutoNEST Server Setup)

"Server License" - The number of permanent licenses purchased.

TRIAL License

"License Limit" - The maximum number of days available to run AutoNEST without a permanent license.

"License Remaining" - The number of remaining days to run AutoNEST without a permanent license.

"Date of Expiry" - This shows the exact expiry date and time of the existing Trial License.

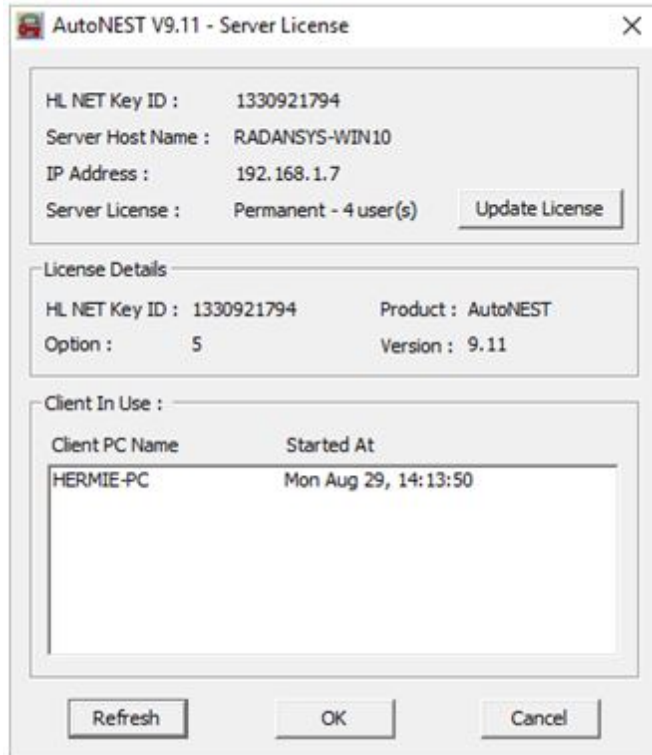
"New Trial License": - This button is used to extend the trial period using a ".V2C" file.

On the Server

Click on the AutoNEST Server License icon



When the **"AutoNEST V9.x Server License"** is clicked, the follow dialog box will appear.



"HL NET Key ID" - This is the unique ID of your usb HL NET key

"Server Host Name" - Host name of your Server (installed with AutoNEST Server Setup)

"IP Address" - IP address of your Server (installed with *AutoNEST* Server Setup)

"Server License" - The number of permanent licenses purchased.



- This button enables you to update your *AutoNEST* license.

License Details

"HL NET Key ID" - This is the unique ID of your usb HL NET key

"Option" - This is the nesting options available in the HL key.

"Product" - Name of product as programmed in the HL key.

"Version" - Version number of the product as programmed in the HL key.

Client In Use

"Client PC" - Name(s) of the PC's that are currently using the *AutoNEST* Server license(s).

"Started At" - The date and time when the client PC takes up one *AutoNEST* Server license.

About AutoNest



This command will display a message box showing the product version number that you are currently using. The following dialog box will be displayed.

You can also double-click “ANEST.VER” on the main folder to find out the VERSION number of the product.



AutoNest File Formats

In order to establish an open architecture for users to have full flexibility in using *AutoNEST*, various file formats must be maintained both at input levels and for output purposes.

Input Files:

For input, there are 2 files required, namely:-

1) **.VEC** file

The **.VEC** file contains the geometry of a part in ASCII file format.

2) **.JOB** file

This ASCII file contains the parameters of a task that need to be nested. For examples, number of stocks, stock size and quantity; number of parts, part quantity and orientation constraints; edge allowance, cutting gap ...etc.

The **3rd file is optional** depending whether there is irregular stock.

3) **.STK** file (optional)

The format of a **.STK** file is the same as the **.VEC** file. Where the **.VEC** describes the profile of a part, **.STK** describes the profile of an irregular stock

4) **.CLS** file (optional – for Cluster-part)

The **.CLS** file contains the relative coordinates of the insertion points of the individual parts in relation to the Cluster's insertion point. Each Cluster requires the individual parts' **.VEC** files, **.VEC** of the Cluster's outline as well as the **.CLS** file.

Output Files:

There are four (4) output files. These files are generated automatically upon a successful nesting. The fifth (DXF) file is generated if the nesting is invoked from Nest Manager instead of in AutoCAD. The sixth file will only be generated if "Common-Line /Toolpath" is invoked.

1) **.SYM** file

This ASCII file gives the layout information by making reference to the Parts position co-ordinates, orientation, color ...etc. and the Stock information.

2) **.SUM** Format

This file contains the nesting summary report in ASCII format.

3) **.XLS** or **.XLSX** Format

This file contains the nesting summary in Excel format.

4) **.XML** file (the contents are similar to **.SYM**)

Two **.XML** files, similar in contents to **.SUM** and **.SYM** are created in Extensible Markup Language.

5) **.DXF** file (created only in the Nest Manager environment)

This file contains the full geometric information of the nested layout(s). As such, it will be very much larger in size when compared to say the **.SYM** file.

6) **.SGD** file

Similar in concept to **.SYM** file except this ASCII file presents the layout information by "partitions blocks" or grids.

VEC Format

Each **.VEC** is a TEXT file containing the x and y co-ordinates of vertices of part profiles arranged in **clockwise** or **counter-clockwise** direction. If some segments of the profiles are arcs, bulge values will be attached to the starting vertices of arcs, or if the profile is a circle, center point and radius of the circle will be recorded. External profile of a part is stored in One example **.VEC** file will be described as follows:-

```

1  #
2  # AutoNEST V9 ENGLISH
3  # Shape Name = test1.vec
4  # First Line = InsertionPoint(x, y), ShapeArea, ShapePerimeter, ShapeRectLength,
5  # ShapeRectWidth, MinRectLength, MinRectWidth, DegreeOfMinRect
6  @ Vec not Compressed
7  9.346454 7.998023 51.850286 48.609869 9.997714 8.402318 10.014353 8.221825 -5.107422
8  9.62808011.478481
9  15.2957988.841770
10 14.6621403.673817
11 7.9735293.076163
12 5.2980848.208960
13 9.62808011.478481

```

```

14  @ Hole 1
15  12.7963707.857398 A 0.388339
16  10.7545837.189432
17  11.2826315.958967 A -0.624107
18  13.2540116.099591
19  12.7963707.857398
20  @ Hole 2
21  9.8040969.580049 C 0.868615
22  @ Hole 3
23  8.7832037.154275
24  8.9944224.798814
25  7.6567006.345684
26  8.7832037.154275
27  @ Leadin 1
28  4.838997 8.555070
29  4.982175 8.555070 A -0.280570
30  5.211919 8.398732
31  5.298084 8.208960
32  5.146448 8.096893 A -0.148813
33  4.951585 8.027744
34  4.817346 8.027744
35  @ Leadin 2
36  7.656700 6.345684
37  7.901215 6.345684
38  @ Leadin 3
39  10.754383 7.1857398
40  13.234154 7.1857398
41  @ Quantity
42  1

```

(Note: The above line numbers are strictly for referencing purposes, they do not appear in the file.)

Description of .VEC File Format

Line 1 to 5 Lines start with "#" character denote comments. There is no limit to the number of comment lines.

The '**AutoNEST V9 English**', must be in one of the comment lines. The '**V9**' reference number is used to check the different formats of (. VEC) files of different software releases. '**English**' indicates what language version of current **AutoNest** you are using.

Line 6 '@ Vec not Compressed' is a file header identifier to indicate this vec part file has not been compressed. It will be followed by a section of an external profile of a part.

Line 7 9.346454 7.998023 51.850286 48.609869 9.997714 8.402318 10.014353 8.221825
-5.107422

First two values are always assumed to be the part's Insertion Point.

The X and Y co-ordinates in real numbers. No restrictions on the length of field.

3rd value is the Part Area.

4th value is the Part Perimeter.

5th & 6th values are the Length & Width of the "Enclosing rectangle" of the part in real numbers.

7th & 8th values are the Length & Width of the "Minimum Enclosing Rectangle" of the part.

9th value is the angle of rotation in order to obtain the "Minimum Enclosing Rectangle".

Line 8-13 X and Y co-ordinates of each vertex of the external profile.

X and Y co-ordinates in real numbers.

No restrictions on the length of field.

Line 14 '@ Hole 1' is file header identifier to indicate the starting of an internal hole profile section of this part

'1' is the hole number of a part.

Line 15-19 'A' is an arc indicator.

12.796370 7.857398 A 0.388339

The first two real numbers are X- and Y- coordinates of starting point of an arc, the last real number is the bulge value of the arc. The ending point of the arc will be the first two real numbers at the following line. If the following line is starting with '@', the ending point will lie in the first line of this section.

This section means the first hole is a polygon with two arcs.

Line 20 '@ Hole 2' indicates the starting of second hole profile section of this part. This hole number is 2.

Line 21 'C' indicates circle.

9.804096 9.580049 C 0.868615

The first two real numbers are X- and Y- coordinates of center point of the circle.

The last real number is radius of the circle. That means the second hole is a circle.

Line 22 '@ Hole 3' indicates the starting of third hole profile section of this part. This hole number is 3.

Line 23-26 A list of X- and Y- coordinates of each vertex of a polygon.

Line 27 "@Leadin 1" indicates the starting of one leadin / leadout section of part profile (in this example, it is for external profile).

Line 28-34 A list of X- and Y- coordinates representing the leadin / leadout lines or arcs. 'A' indicates Arc - same as the foregoing.

Line 35-40 Other leadin / leadout for part inner profiles.

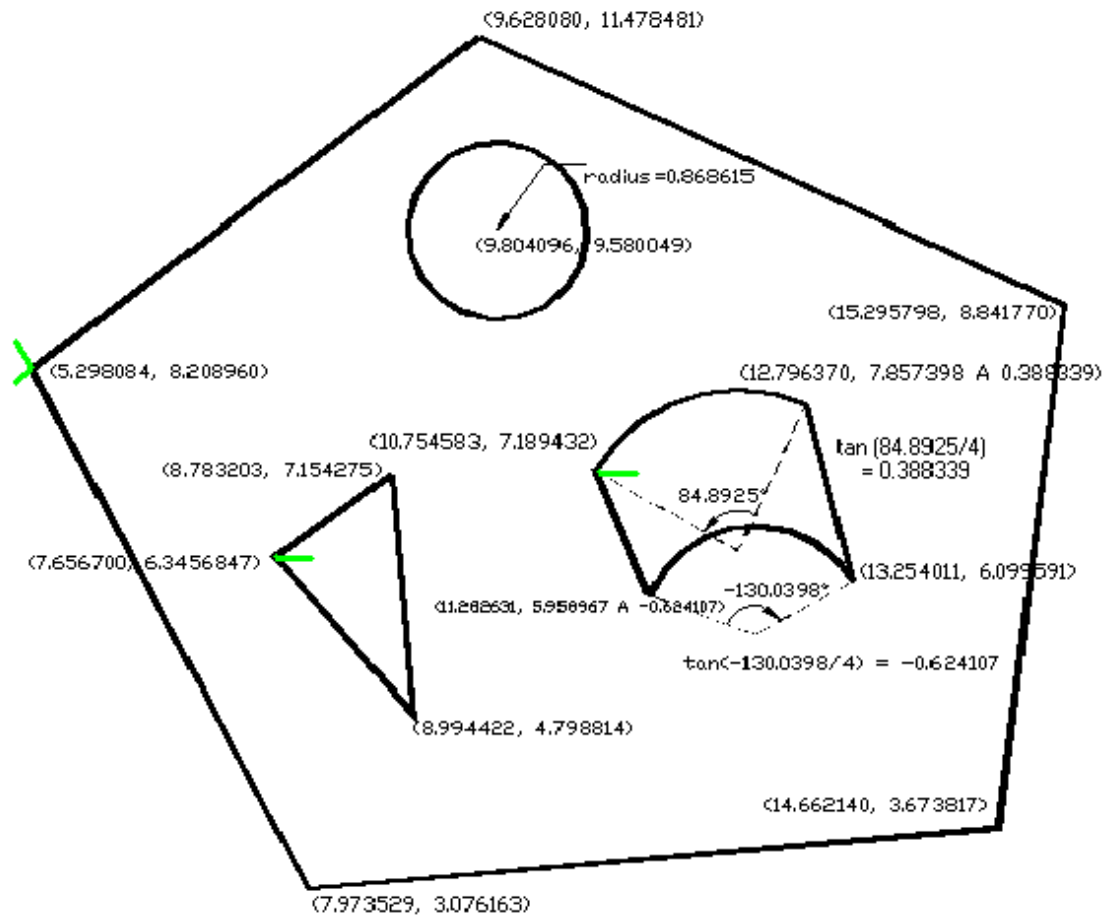
Line 41 "@ Quantity" is file header identifier for part quantity. This is optional.

Line 42 Quantity of the part. (optional)

Notes on overall geometry of Part

- Each profile of geometry must be closed.

- No crossing over on each profile itself or between profiles.
- Not more that 1500 vertices per profile, which includes the starting and ending vertices of arcs.



.STK File Format

.STK is identical to .VEC in format. Each **.STK** is a TEXT file containing the x and y co-ordinates of vertices of an irregular stock (ir-stock) profile(s) arranged in **clockwise** or **counter-clockwise** direction. If some segments of the profiles are arcs, bulge values will be attached to the starting vertices of arcs, or if the profile is a circle, center point and radius of the circle will be recorded. One example of a **.STK** file is described as follows:

```

1  #
2  # AutoNEST V9 ENGLISH
3  # Shape Name = i-stk5.vec _anest_.dxf
4  # First Line = InsertionPoint(x, y), ShapeArea, ShapePerimeter,
5  # ShapeRectLength, ShapeRectWidth, MinRectLength, MinRectWidth,
6  #
7  @ Vec not Compressed
8  1348.368819 1166.007848 9300121.582944 13124.084835 3806.703954 3258.750718
9  76.849897 3491.494663
10 1552.213191 3464.786885
11 1685.943538 3037.462253
12 2207.491994 2182.812964

```

```

13 3183.723608 1969.150661
14 3883.553851 1555.179906
15 3883.553851 232.743945
16 77.681652 232.743945
17 76.849897 3491.494663
18 @ Quantity
19 1

```

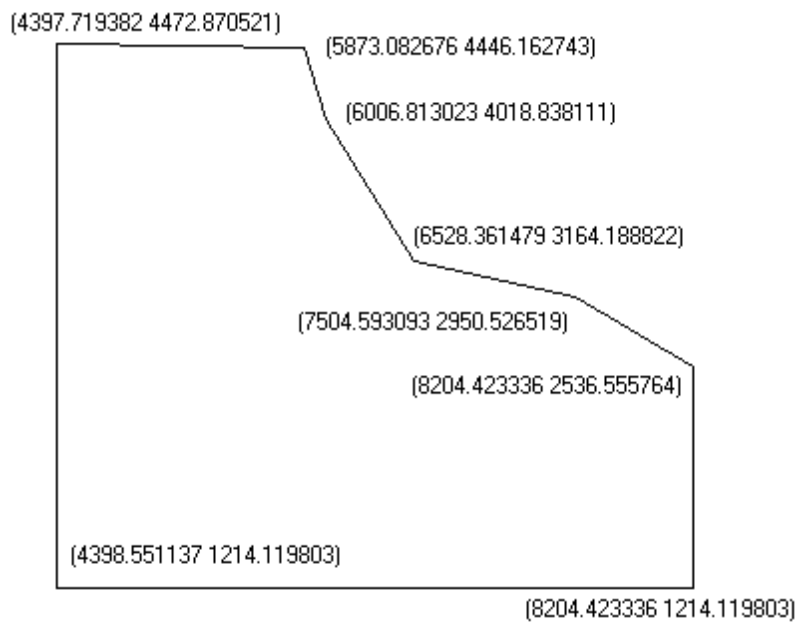
(Note: The above line numbers are strictly for referencing purposes, they do not appear in the file.)

Description of .STK File Format

- Line 1 to 6** Lines start with "#" character denote comments. There is no limit to the number of comment lines.
The '**AutoNEST V9 English**', must be in one of the comment lines. The '**V9**' reference number is used to check the different formats of files of different software releases.
'**English**' indicates what language version of current *AutoNest* you are using.
- Line 7** '**@ Vec not Compressed**' is a file header identifier to indicate this ir-stock file has not been compressed. It will be followed by a section describing the external profile of an ir-stock.
- Line 8** First pair of co-ordinates is always assumed to be the ir-stock's Insertion point. X and Y co-ordinates in real number. No restrictions on the length of field.
Second real number is the Area of ir-stock.
Third real number is the Perimeter of ir-stock.
The last two real numbers are the Length & Width of the enclosing Rectangle of the ir-stock.
- Line 9-17** X and Y co-ordinates of each vertex of the external profile.
X and Y co-ordinates in real numbers.
No restrictions on the length of field.
- Line 18** "**@ Quantity**" is file header identifier for stock quantity. This is optional.
- Line 19** Quantity of the irregular stock sheets. (optional)

Notes on overall geometry of an Irregular stock

- Each profile of geometry must be closed.
- No crossing over on each profile itself or between profiles.
- Not more than 1500 vertices per profile, which includes the starting and ending vertices of arcs.

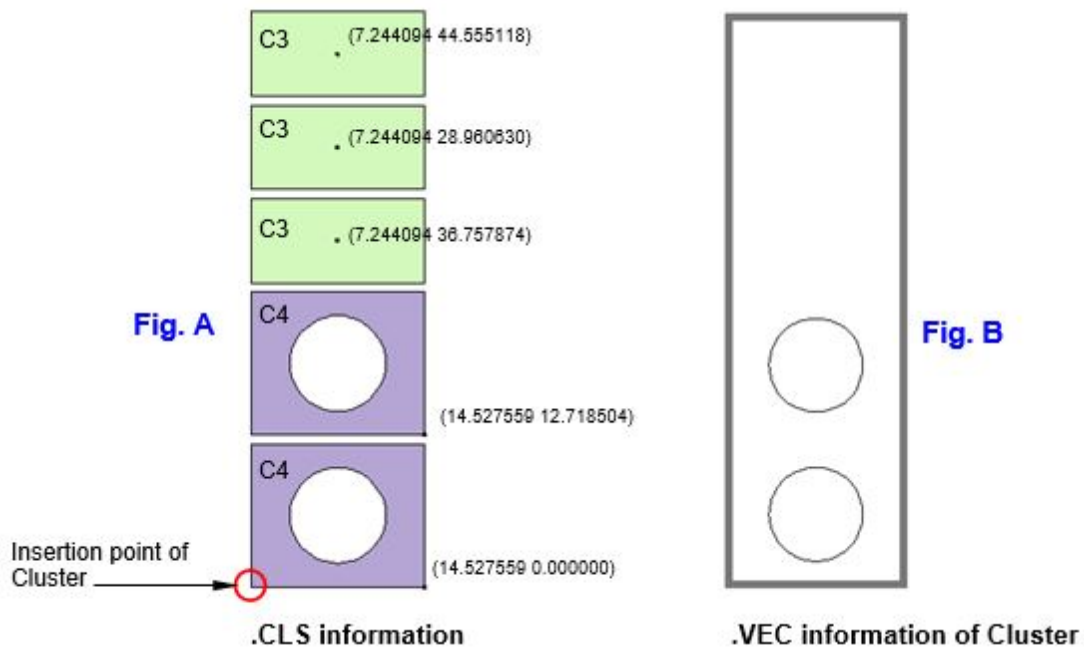


.CLS File Format

For each Cluster, the following files are required :-

- .VEC of the individual parts
- .VEC of the Cluster
- .CLS of the Cluster

Take for example, the Cluster-part below which consists of 3 nos of Part C3 and 2 nos of Part C4. The .CLS and .VEC of the Cluster is described below.



.CLS file contains the relative co-ordinates and angle of each of the individual part's position to the Cluster's insertion point. (See Fig. A above)

```

1  #
2  # AutoNEST V9 ENGLISH
3  # Cluster Name = ClusterName.cls
4  # First Line = Part-name, Relative-X-coordinate, Relative-Y-coordinate, Rotation-Angle
5  #
6  C3 7.244094 44.555118 0.000000
7  C3 7.244094 28.960630 0.000000
8  C3 7.244094 36.757874 0.000000
9  C4 14.527559 12.718504 90.000000
10 C4 14.527559 0.000000 90.000000

```

(Note: The above line numbers are strictly for referencing purposes, they do not appear in the file.)

Description of .CLS File Format

Line 1 to 5 Lines start with "#" character denote comments. There is no limit to the number of comment lines.

The '**AutoNEST V9 English**', must be in one of the comment lines. The '**V9**' reference number is used to check the different formats of files of different software releases. '**English**' indicates what language version of current *AutoNest* you are using.

Line 6 to 10

- 1st parameter : Part name
- 2nd parameter : Relative x-ordinate of Part's insertion point to Cluster's insertion point.
- 3rd parameter : Relative y-ordinate of Part's insertion point to Cluster's insertion point.
- 4th parameter : Rotation angle of Part in degrees

.VEC file of a Cluster contains the enclosing rectangle of the Cluster or user-defined outline of the Cluster and the internal holes information. (See Fig. B above)

```

1  #
2  # AutoNEST V9 ENGLISH
3  # Shape Name = Cluster.vec_anest_dxf
4  # First Line = InsertionPoint(x, y)
5  #
6  @ Vec not Compressed
7  1958.630432
3109.863868
8  1958.630432
3109.863868
9  1958.630432
3157.942608
10 1973.157991 3157.942608
11 1973.157991 3109.863868
12 1958.630432 3109.863868
13 @ Hole 1
14 1965.894211 3128.566624 C
4.000000
15 @ Hole 2
16 1965.894211 3115.848120 C 4.000000
17 @ Quantity
18 1

```

Enclosing Rectangle/User-defined Cluster outline information
 1st line - co-ordinates of the Insertion Pt.
 Subsequent lines are the co-ordinates of the Enclosing rectangle of the Cluster

Internal holes of Cluster (see Fig. B on the pervious page)

.JOB File Format

A **.JOB** file will contain all the relevant controlling parameters of the nesting for a particular task – as captured in the dialog boxes when **TaskEdit** is invoked. The format of the file is as follows:

```

1  #
2  # AutoNEST V9 ENGLISH
3  # Job Name = EXAMPLE1
4  # Next 6 lines = No of Distinct Shapes, No of Distinct R-Stock & I-Stock,
5  # Cutting Gap, Edge Allowance X and X1,
6  # Edge Allowance Y and Y1 Control Parameter
7  4
8  2 2 1
9  5.0000 0.0000
10 10.0000 10.0000
11 10.0000 10.0000 0.0000
12 C R 1 3 0 2 1 0 600.0000 600.0000 2 15.0000
13 1200.0000 x 2400.0000 10 1 77.00 0 0.00
14 1500.0000 x 3000.0000 10 1 99.00 0 0.00
15 &I-STK 1 1 Y
16 &I-STK5 1 1 Y
17 20 W17 (0 90 180) Y 1 W17 0 1 5
18 20 W18 (ALL) Y 1 W18 0 1 5
19 10 W20 (step 10 0 180) Y 1 W20 0 1 5
20 20 W23 (step 1 0 90) Y 1 W23 0 1 1
21 BRIDGE 5.0000 20.0000 50.0000 4
22 SKELETON-CUT 850.0000 750.0000 3.0000 1

```

(Note: The above line numbers are strictly for referencing purposes, they do not appear in the file.)

IMPORTANT: Do not use 'Tab' characters when you are constructing this file with a text editor or word processor. Instead use ordinary spaces.

Description of .JOB File Format

Line 1 to 6 Lines starting with "#" are comments. There is no limit to the number of comment lines. The '**AutoNEST V9 English**' must be in one of the comment lines. The '**V9**' reference number is used to check the different formats of (.JOB) files for different software releases. '**English**' indicates what language version of current *AutoNest* you are using.

Line 7 Number of distinct parts (Range: 1 to 1000)

Line 8 The 1st digit represents the number of distinct regular stocks
Range : 1 to 5000 (max. value as set in ANEST.SYS)

The 2nd digit represents the number of distinct irregular stocks
Range : 1 to 5000 (max. value as set in ANEST.SYS)

The 3rd digit represents "Multi-Stock Control"
0 - means "Auto"
1 - means "Best Yield"
2 - means "Lowest Cost"

"**Auto**" is the default setting and is the fastest option where the nesting engine will appraise the parts and stocks and then automatically selects the largest suitable Stock to nest.

"**Best Yield**" - in this option the nesting engine will iterate through each Stock size to nest the available parts. For each nested Stock, it will compare the nested utilization area of Stock and then select the one with the highest utilization. It will continue to iterate until all the parts or stocks are used up.

"**Lowest Cost**" - in this option the nesting engine will iterate through each Stock size to nest the available parts. For each nested Stock, it will compare the Nested Area per Cost of Stock and then select the one with the highest Nested Area per unit cost.

For this option to work, the "Cost per Stock" must be specified.

Both the "Best Yield" and "Lowest Cost" will take a longer time to nest.

*****The more the number of distinct Stocks, the longer it will take to nest for "Best Yield" or "Lowest Cost" options*****

If either "Best Yield" or "Lowest Cost" option is selected, the Stock Priority will be ignored.

Line 9 1st parameter denotes "Cutting gap" (Max field length : 14 real nos.)
2nd parameter denotes "Shear Blade Limit".
(Max field length : 14 real nos, >=0.0) If this value is "0" - it means NO shear nesting required;
if greater than "0" then it means "Yes" to shear nesting.

Line 10 Edge allowance [x] and [x1] (Left and right side of the regular stock)
Max field length: 14 (real nos.)

Line 11 Edge allowance [y] and [y1] (Bottom and top of the regular stock) and Edge allowance of irregular stock.
Max field length: 14 (real nos.)

Line 12 Nesting control parameters.

The 1st parameter can be:

'D' - means single part nesting with highest Density (in single array)

'M' - means single part nesting with Maximum quantity (in single array)

'C' - means single part nesting with Combination of density & maximum qty (in mixed array)

'E' - means extension - long nest.

The 2nd parameter is to specify which type of stock to be used first if there are both Regular Stock and Irregular Stock in the same job/ task.

'R' - means nest all Regular Stocks first before any Irregular Stocks

'I' - means nest all Irregular Stocks first before any Regular Stocks.

The 3rd parameter is for Packing Start Point

- 1 Left Bottom
- 2 Left Top
- 3 Right Top
- 4 Right Bottom

The 4th parameter is for Packing direction control

- 1 Horizontal Packing
- 2 Vertical Packing
- 3 Auto (System Control)
- 4 Horizontal Packing for All Stocks
- 5 Vertical Packing for All Stocks

The 5th parameter is for Common Line option

- 0 Without Common Line Consideration
- 1 With Common Line Consideration

The 6th parameter is for Mirror option (Single part only)

- 0 Do NOT allow Part Mirror
- 1 Allow Part Mirror
- 2 Nest Half Quantity with Mirrored Parts

The 7th parameter is for Ignore Part Hole option

- 0 Ignore 'Hole' of part (will not nest any parts inside part holes)
- 1 Do NOT ignore 'hole' of part (will nest smaller parts inside part holes)

The 8th parameter is for Save Remnant option

- 0 Do NOT save remnant stocks
- 1 Save remnant stocks into .STK files.

The 9th and 10th parameters are for Min. Remnant Size in X and Y. Only remnant stocks that are greater than this Min. Remnant Size will be saved into .stk files onto the default Part/ Ir-Stock directory and provided the “Save Remnant” option (8th parameter) is set to 1.

By default these remnants will be saved onto the Part/ Irregular Stock directory. The names of the remnant stocks will be *Taskname_remnant1-1.stk*, *Taskname_remnant2-1.stk*, *Taskname_remnant2-2.stk* and so on. The naming convention is as follows :- *_remnant(LAYOUT_NO)-(NO) where “remnant” is the default name as defined in the 6th parameter of @REMNANT_CONTROL in ANEST.SYS.

In addition, a Task will automatically be created using the default name ‘*Taskname_rem.job*’ onto the Task directory.

The 9th and 10th parameters are for Minimum Remnant Size in X and Y. Only remnant stocks that are greater than this Min. Remnant Size will be saved into .stk files onto the default Part/ Ir-Stock directory and provided the “Save Remnant” option (8th parameter) is set to 1.

The 11th and 12th parameters are for ‘Parts Priority Control’.

11th parameter is for “Max. No. of Priority per Stock” (Integer >=0)

12th parameter is for “Min. Nest-able Remaining Space” percentage. (real nos.)

“Max. No. of Priority per Stock” enables you to limit the no. of Parts of different priorities to be nested into a stock.

This is to avoid the confusion on the production floor where the operator, after cutting has to sort the parts and placed them into their respective “carts”.

“Minimum Nest-able Remaining Space” percentage – is the parameter for the remaining space after ALL the parts of the specified Priority have been nested onto the stock.

If 0 is defined, then any remaining space is considered “nest-able”, AutoNEST will try to find suitable parts to fit onto the remaining space.

If 10 is defined, the remaining space must be >=10% of the stock size to be considered “nest-able”, AutoNEST will then try to find suitable parts to fit onto the remaining space.

If 100 is defined, then any remaining space is considered NOT “nest-able”, AutoNEST will not attempt to nest into the remaining space after ALL parts of the specified priority have been nested.

Line 13-14 A list of distinct regular stocks /plates. Only the first 5 parameters are mandatory.
Each line has the following format:

Stock-Width x Stock-Length Quantity Priority Cost-per-Stock 0 0.0 0 0.0 No-of-Small-Part-Zones X-Y-Coordinates-of-SP-Zone1 X-Y-Coordinates-of-SP-Zone1 X-Y-Coordinates-of-SP-Zone2 X-Y-Coordinates-of-SP-Zone2

The first 2 parameters are for Width x Length of stock plate or sheet.

3rd parameter is the Quantity of the stock plate /sheet.

4th parameter is the Priority of the stock plate /sheet. 1 has the highest priority whereas 99 has the lowest priority. This refers to the priority for regular stocks only.

5th parameter is the cost per stock plate /sheet.

6th & 7th parameters are currently used in AutoNEST FX (not applicable here)

8th & 9th parameters are newly added parameters for future use.

10th parameter : No. of Small-Part Zones (1 to 4)

11th parameter : X & Y co-ordinates of the bottom left corner of Small-Part Zone 1 with reference to the bottom-left corner of the sheet.

12th parameter : X & Y co-ordinates of the upper right corner of Small-Part Zone 1 with reference to the bottom-left corner of the sheet.

13th parameter : X & Y co-ordinates of the bottom left corner of Small-Part Zone 2 (IF APPLICABLE) with reference to the bottom-left corner of the sheet.

14th parameter : X & Y co-ordinates of the upper right corner of Small-Part Zone 2 (IF APPLICABLE) with reference to the bottom-left corner of the sheet.

Line 15 - 16 A list of distinct irregular stocks (ir-stock). Each line has the following format:

&ir-stock Quantity Priority Rotate-able

'&' is a prefix to differentiate the irregular stock from the regular stock.

ir-stock is the .STK file name (ir-stock). The .STK file stores the ir-stock geometry profile.

Quantity refers to the quantity of the irregular stock.

Priority of the ir-stock to be nested. 1 has the highest priority whereas 99 has the lowest priority. This refers to the priority for the irregular stock.

Rotate-able has two options. "Y" means able to rotate the ir-stock and "N" means no rotation of the ir-stock.

Line 17-20 A list of distinct Parts.

Number of lines depends on number of distinct Parts. Each line has the following format:

Part-quantity Part-name (Orientation constraints) Pairing Priority Block-name Filler-quantity Support+ 180 Step-angle

Part-quantity - integer number (range: 1 - 9999)

(Note: for Single Part tasks of fixed size stocks, part-quantity can be 'M' to indicate unlimited quantity of the Part)

Part-name - string, 30 characters long (no space characters allowed)

Orientation constraints - there are 2 formats to define orientation constraints. Both are acceptable and can co-exist in the same file.

Format A

Combination of any angles between 0 and 360 (separated by SPACE such as :-
(30 60 90 180 ...) or (ALL))

Format B

This new format is helpful if the Orientation angles consist of a long list and if the angles increase by a constant value.

(step 10 0 180)

where:

step is the reserved word to denote "increment angle"

10 is the increment angle (must be greater than 0 and a positive value)

0 is the Start Angle (real nos., range -360 ~ +360)

180 is the End Angle (real nos, must be greater than Start Angle)

If represented in Format A

(0 10 20 30 40 50 60 70 80 90 100 110 120 130 140180)

Pairing - 'Y' means allow part pairing. 'N' means do not allow part pairing. Currently, not in use.

Priority - Priority of the parts to be nested. 1 has the highest priority, 2 has the second highest and so on.

Block-name - Part details block name. Part detail block is a DWG file containing more detail information of the part. In the current version, it will be the same as Part-name.

Filler-quantity - integer number (range: 1- 9999). Additional quantity for filling up available space.

Support+180 - 1 means allow the part orientation angle + 180 degree. Eg. if part allows 30 degree orientation, 30 + 180 = 210 degree will also be allowed.

0 means "No" to "+180" degrees for part orientation.

Step-angle - an incremental step angle for seeking the best nesting pattern. Reducing the step-angle will have higher chance to get a better nesting result but it will also increase the nesting time.

Line 21 Auto-bridge parameters

The 1st parameter indicates starting of Auto-Bridge parameters.

The 2nd parameter is the bridge width.

The 3rd parameter is the minimum corner offset.

The 4th parameter is the minimum angle (range 45~90 deg).

The 5th parameter is the maximum number of bridges per part.

(See below for illustration of these parameters)

Line 22 Skeleton-Cut-Off parameters

The 1st parameter is the X-Spacing or No. of X equal spacings.

The 2nd parameter is the Y-Spacing or No. of Y equal spacings.

The 3rd parameter is the offset distance from part profile or skeleton-bridge.

The 4th parameter : 1 indicates that 1st & 2nd parameters are real values.

2 indicates that 1st & 2nd parameters are actually no. of equal spacings for X and Y respectively, not the real values.

(See below for the illustration of Skeleton-Cut-Off parameters)

5th parameter – Minimum area of Scrap-pocket

6th parameter – Maximum area of Scrap-pocket

7th parameter – Minimum length of Scrap-pocket

Illustration for Auto-Bridge Parameters

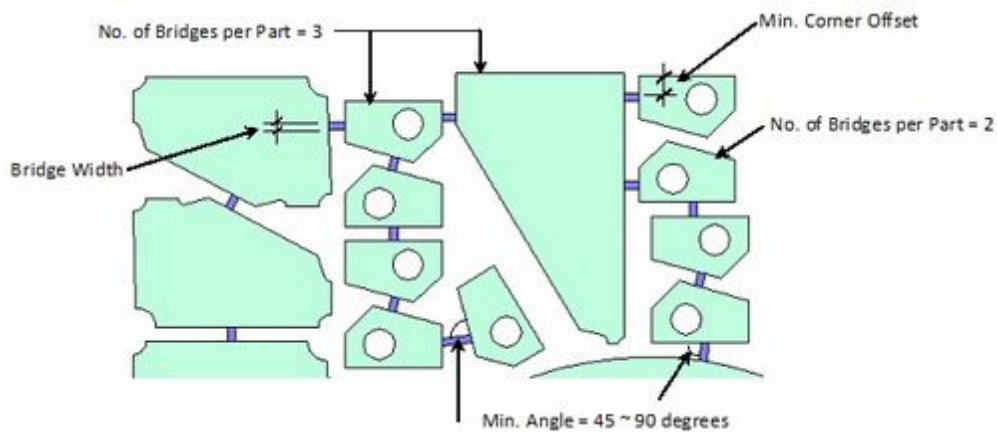


Illustration of Scrap-Pocket

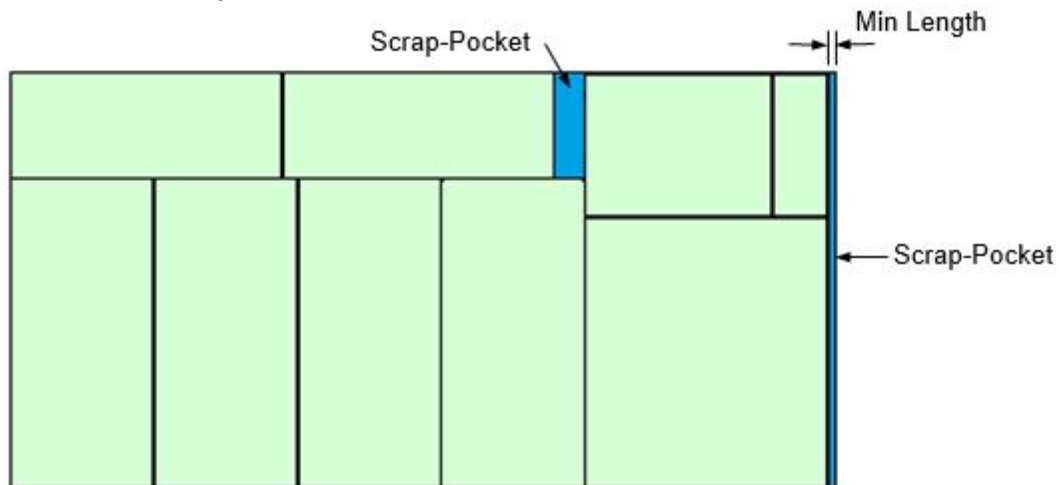
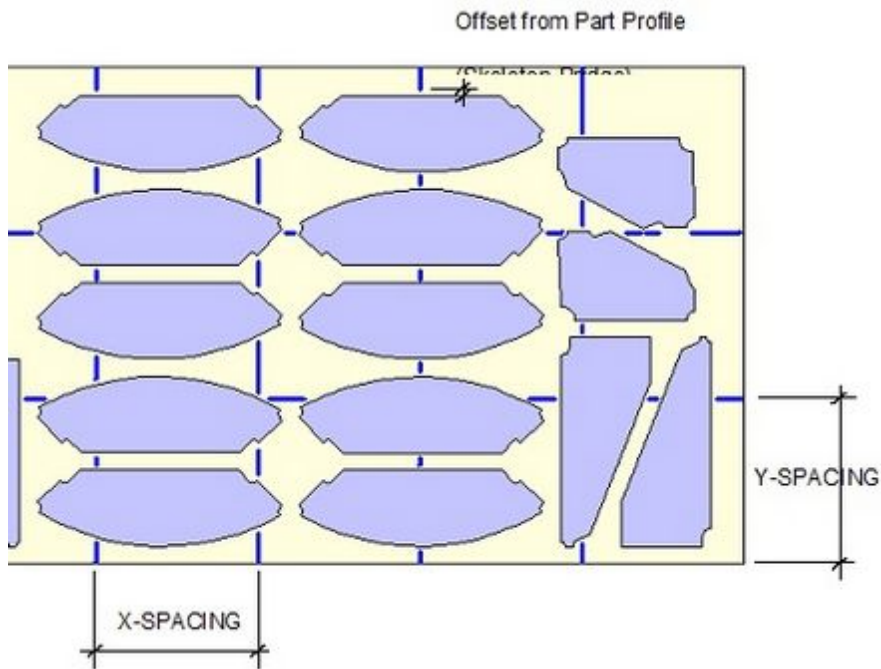


Illustration for Skeleton-Cut-Off Parameters



.SYM File Format

All the pertinent information of the nested results are in a file with the filename extension of **.SYM**. Assuming that the task name is *example*, it will therefore be *example.sym*.

The **.SYM** file is intended to be the standard vehicle for communicating with any other applications.

From AutoNEST V9.5.3 onwards, the .SYM file format has been changed to V9 REV1. (previously it was "V9"). See below :-

The format is as follows:

```

1  #
2  # AutoNEST V9
3  # Sym File Name = EXAMPLE.sym (REV1)
4  # Layout from NESTPRO.EXE
5  #
6  # Process Time Taken
7  #
8  # Input: 0 min 0.98 sec
9  # Pair I: 0 min 0.79 sec
10 # Pair II: 0 min 0.06 sec
11 # Packing: 0 min 0.18 sec
12 # Total: 0 min 2.03 sec
13 #
14 JOB = BIGPART
15 No of Distinct Shapes = 8
16 Total No of Shapes = 8
17 Total No of Stock Sheet = 3 0
18 Encl Rect = (15.000000 15.000000) (8883.888560 3018.000000)
19 Stock Sheet = (3048.000000 9144.000000) x 2 99.00

```

20 *Sum of Area of Shapes = 18935546.178672*
 21 *(BW4 1399.477698 717.433929 0.000000 5 0 1)*
 22 *(BW3 7386.392422 2313.960112 0.000000 4 0 1)*
 23 *Encl Rect = (15.000000 15.000000) (8830.763448 3018.000000)*
 24 *Stock Sheet = (3048.000000 9144.000000) x 1 99.00*
 25 *Sum of Area of Shapes = 22917695.314072*
 26 *(BW2 1099.802730 2029.470929 180.000000 3 0 1)*
 27 *(BW1 8067.357744 1106.283027 180.000000 2 0 1)*
 28 *Encl Rect = (15.000000 15.000000) (7744.557323 3018.000000)*
 29 *Stock Sheet = (3048.000000 9144.000000) x 1 99.00*
 30 *Sum of Area of Shapes = 16455773.523961*
 31 *(BW9 2244.239528 2501.755357 180.000000 3 0 1)*
 32 *(BW8 2146.288772 607.457250 0.000000 2 0 1)*
 33 *(BW5 5603.085323 2510.045662 0.000000 6 0 1)*
 34 *(BW7 5663.686544 537.954338 180.000000 1 0 1)*

(Note: The above line numbers are strictly for referencing purposes, they do not appear in the file.)

Description of .SYM File Format

- Line 1 to 13** Lines start with "#" character denote comments. There is no limit to the number of comment lines.
The '**AutoNEST V9**' must be in one of the comment lines. The '**V9**' reference number is used to check the different formats of (.SYM) files for different software releases.
- Line 14** Sym file name.
The key character here is the "=" equal sign. The words before it are purely descriptive but the name after the sign is important.
- Line 15** Number of distinct parts.
The key character here is the "=" equal sign. The words before it are purely descriptive but the value after the sign is important.
- Line 16** Total quantity of parts nested.
The key character here is the "=" equal sign. The words before it are purely descriptive but the value after the sign is important.
- Line 17** Total number of distinct nested layouts.
The key character is the "=" equal sign. The words before it are purely descriptive but 2 values (separated by space) indicate the no. of distinct nested layouts for regular and irregular stocks respectively.
- Line 18-22** The first nested layout information.
- Line 18** Enclosing rectangle of the first nested layout.
The first pair of real numbers is the left-bottom point of the rectangle, the second pair is the length and width of the rectangle.
- Line 19** Stock sheet size and the number of the repeated layout and Cost per stock.
Within brackets are the width and length of the stock.
The last number is the Cost per stock.
- Line 20** Sum area of parts nested in the current layout.

Line 21-22 A list of nested parts. Each line has the following format:

(Part-name X Y Angle Color Hole_no Layer)

Part name - name of the part

(X, Y, Angle) - Position of the part, relative to the left bottom corner of the respective stock.

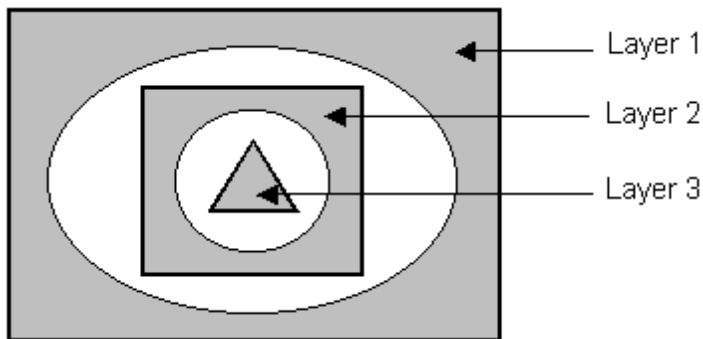
Color - indicate the color to be used on screen

Hole-no - indicate which hole the current part is being nested as the Part may have multiple holes (these are differentiated by the "Hole no." in **.VEC** file

'0' means the part is NOT inside any other part's hole.

'>0' means the part is inside a certain hole no. of the part

Layer - the meaning is different from the Drawing layers that most of us are familiar with. Here, it means whether a part is nested in multiple levels of "Part-In-Part". The diagram below illustrates what it means to be called "Layer 1", "Layer 2", "Layer 3" and so on.



Line 23-34 The second and third nested layout information.

The part X/Y coordinates and the enclosing rectangle in the second layout are relative to the **left**

bottom corner of the 2nd stock.

Similarly, the part X/Y coordinates and enclosing rectangle in third layout are relative to the **left**

bottom corner of the 3rd stock and so on

IMPORTANT: Do not use 'Tab' characters when you are constructing this file with a text editor or word processor. Instead use ordinary spaces.

An **Irregular stock** will have the following **.SYM** file format :-

```

1  #
2  # AutoNEST V9
3  # Sym File Name = IRREG-1.sym
4  # Layout from NESTPRO.EXE
5  #
6  # Process Time Taken
7  #
8  # Input: 0 min 0.08 sec
9  # Pair I: 0 min 0.68 sec
```

```

10 # Pair II: 0 min 0.02 sec
11 # Packing: 0 min 1.98 sec
12 # Total: 0 min 2.76 sec
13 #
14 JOB = IRREG-1
15 No of Distinct Shapes = 5
16 Total No of Shapes = 87
17 Total No of Stock Sheet = 0 3
18 Encl Rect = (0.000000 0.000000) (3806.703954 3258.750718)
19 Stock Sheet = (3258.750718 3806.703954) x 1
20 Stock Area = 9234599.16 Sum of area of shapes = 5673622.50
21 (&i-stk 2685.912593 1921.541344 0.0 1)
22 (17-1 3445.292074 1876.218412 11.766537 2 0 1)
23 (17-1 2854.195540 2200.574867 -168.233463 2 0 1)
24 (17-1 2163.516778 1875.891255 11.766537 2 0 1)
25 (17-1 1572.420244 2200.247710 -168.233463 2 0 1)
26 (17-1 418.622195 2774.246774 -269.985376 2 0 1)
27 (17-1 2566.064366 443.120389 -179.985376 2 0 1)
28 (17-1 2398.265618 961.253189 -179.985376 2 0 1)
29 (PART3 2117.604589 1421.560227 0.014624 3 0 1)
30 (PART3 2324.528435 1260.959391 180.014624 3 0 1)
31 (PART3 3624.289236 376.923330 267.808871 3 0 1)
32 (PART3 1849.929902 1349.789402 270.014624 3 0 1)
33 (PART3 2851.255095 1352.473314 180.014624 3 0 1)
34 (PART4 154.152797 3096.646617 270.000000 4 0 1)
35 (W19 994.017396 2813.313029 180.014624 5 0 1)
36 (W19 928.893289 1829.261109 0.014624 5 0 1)

```

The difference between regular stock and irregular stock are as follows :-

- Line 19** Overall dimension of the irregular stock and the number of repeated layout(s). (Within brackets are the overall width and length)
- Line 20** Area of irregular stock and Sum of area of parts nested in the current layout.
- Line 21** List ir-stock name and position. The format for this line is:
&ir-stock-name X Y Angle Color
'&' is a prefix to differentiate the irregular stock from the regular stock.
ir-stock-name is the .STK file name of the ir-stock.
X, Y, Angle - transformation of the ir-stock, relative to its enclosing rectangle's left bottom point (0, 0).
 In this current version the **Angle** is always 0.0. and the **Color** always 1.

Transformation of SYM File

IMPORTANT NOTES

From this version V9.5.3 onwards, there is a change in the SYM file. The ONLY difference between this new format and the previous is the **X/Y co-ordinates** reference point of transformation of the PARTS. This SYM format is named "V9 REV1".

Under the new SYM format, each layout uses its own stock rectangle's **left-bottom point** as the reference point or starting point, with the coordinate of (0,0). Each part's X/Y data or co-ordinates in the SYM file is relative to its respective stock rectangle's **left-bottom point**. (It is no longer relative to the first layout's starting point – as in previous version) Under the new SYM format, Parts, Stock Sheets and Enclosing Rectangles are all transformed in the same way, through a set of 'external or predefined settings' in the **ANEST.SET** file, under the heading @LAYOUT. See below :

@LAYOUT

```
1.1 0    # Horz display ratio between layouts (0:based on individual stock length; 1:based on
         longest stock length)
40       # Report length of layout summary
1.2      # Vert display ratio between rows (based on the biggest stock width)
0        # Max no. of layouts displayed per row (0:no limit)
1        # Location of layout summary (1:right; 2:top-left; 3:top-right)
```

For a detailed explanation of the settings in ANEST.SET, @LAYOUT, please refer to Chapter 7 pages '7-22' – '7- 24'.

See Page '6-25' for an illustration of the new .SYM file format ("V9 REV1").

Parts Referencing

In the first nested layout, the X/Y co-ordinates of each part is relative to the **Insert Pt** (the **left-bottom point of 1st stock**). **Insert Pt** is obtained interactively through AutoCAD as one of the prompts during the *Nesting* command. If running in the Nest Manager environment (Windows), a default value of (0, 0) is taken, even though users can still set a different value in the ANEST.SET file.

To display the nested parts in the 2nd layout, users can predefine the offset distance from the first layout. By default, a distance of '1.1' ratio of the Stock Length is used. This means the second layout is positioned at an offset distance of '1.1 x 1st Stock Length'.

$$(1.1 \times 1^{\text{st}} \text{ Stock Length})$$

Users can change this ratio via the 1st parameter in @LAYOUT (ANEST.SET). If 'Layout Summary Report' is chosen to be displayed also, the offset distance from first layout to second will be :-

$$(1.1 \times 1^{\text{st}} \text{ Stock Length}) + (40 \times \text{Textsize of "Layout Summary report"})$$

where "1.1" & "40" are defined in @LAYOUT (ANEST.SET).

Textsize of "Layout Summary report" is defined in Sysdata (8th line in ANEST.ARG)

The same transformation rule applies to the display of the subsequent nested layouts.

Based on the above information, OEM Partners can make their nested layouts transformation according to their own application needs. Now the X/Y coordinates in the SYM file are all 'absolute' data (no offsets included) and the various offsets can be preset externally in ANEST.SET – thus making the work if any, easier for OEM Partners.

Stock Sheet Referencing

The first stock sheet (rectangle) is drawn or displayed from the left bottom – which is the Insert Pt. Subsequent stock sheets are drawn with the specified offset distance as set in @LAYOUT in ANEST.SET (refer to 1st and 3rd parameters)

Enclosing Rectangle Referencing

No change from the previous version.

Enclosing Rectangle is the minimum rectangle that encloses the nested parts within the stock. The enclosing rectangle coordinates in SYM file is in this format: **(X Y) (Length Width)**, where (X Y) is the location of its left bottom point which is relative to the left bottom point of its respective **stock sheet** rectangle (not the original **Insert Pt.**). The transformation of the Enclosing Rectangle is the same as Stock Sheet and Nested Parts.

Repeated Layouts

No change from the previous version.

Note that in some cases, there can be repeated layouts, i.e. nesting patterns with identical arrangement of nested parts and quantities. In the SYM file, where there are “repeated layouts” it is indicated on the following lines (based on the .SYM file in page ‘6-18’):

19	<i>Stock Sheet = (3048.000000 9144.000000) x 2</i>	{ 2 nos of identical layouts}
24	<i>Stock Sheet = (3048.000000 9144.000000) x 1</i>	{ only 1}
29	<i>Stock Sheet = (3048.000000 9144.000000) x 1</i>	{ only 1}

.SUM File Format

The **.SUM** file is one of the three files created upon every successful nesting of a task.

A **.SUM** file will contain almost the same information as the Nesting Summary report when displayed on the drawing. It will display the Part Area of each parts whereas the Summary report in AutoCAD will not. The format is listed below:

NESTING REPORT OF TASK NAME : BIGPART

STOCK QTY	STOCK #1 1	STOCK #2 1	STOCK #3
STOCK SIZE	3048.00x9144.00	3048.00x9144.00	3048.00x9144.00
ENCLOSING RECT	3018.00x8883.89	3018.00x8830.76	3018.00x7744.56
ENCLOSING RECT/ STOCK AREA	26651244.09/ 27870912.00 95.62%	26811575.67/ 27870912.00 96.20%	23373074.00/ 27870912.00 83.86%
TOTAL PART AREA ENCLOSING RECT	22917695.31/ 26651244.09 85.99%	18935546.18/ 26811575.67 70.62%	16455773.52/ 23373074.00 70.40%
STOCK UTILIZATION	82.23%	67.94%	59.04%
TOTAL PART PERIMETER	34744.63	40192.03	44050.47
PART NAME			
BW1	1/1	0/1	0/1
BW2	1/1	0/1	0/1
BW3	0/1	1/1	0/1
BW4	0/1	1/1	0/1
BW5	0/1	0/1	1/1
BW7	0/1	0/1	1/1
BW8	0/1	0/1	1/1
BW9	0/1	0/1	1/1
SUB TOTAL	2/8	2/8	4/8

NESTING SUMMARY

PART NAME	BASIC QTY	FILLER QTY	PART AREA
BW1	1/1	-	11458847.66
BW2	1/1	-	11458847.66
BW3	1/1	-	9466031.98
BW4	1/1	-	9469514.19
BW5	1/1	-	4117914.30
BW7	1/1	-	4113026.03
BW8	1/1	-	4116085.41
BW9	1/1	-	4108747.79

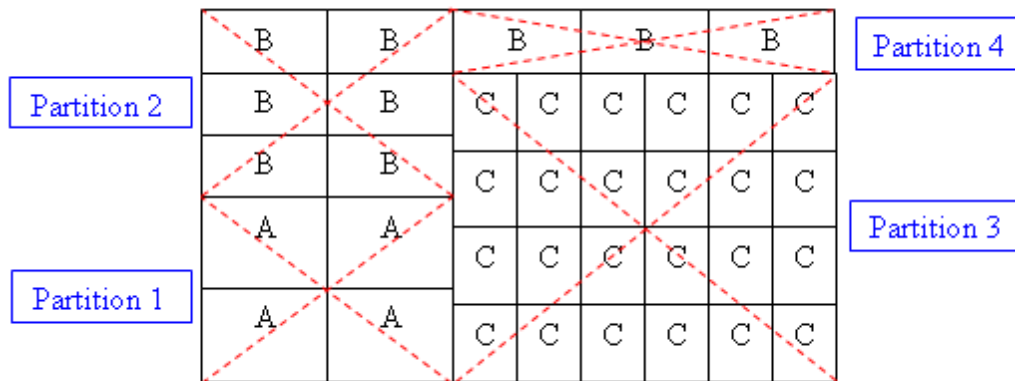
TOTAL PARTS NESTED 8/8 -

STOCK	QTY	COST
3048.00x9144.00	3	2400.00
TOTAL	3	2400.00

.SGD File Format

If "Toolpath" or "common-line" is invoked or if the "Nest rectangle parts in regular grids" at @NEST_CONTROL (set in the ANEST.SYS) is set to "1", an extra file with the filename extension of **.SGD** will be generated after nesting. **.SGD** will be generated besides **.SYM** and **.SUM**. Assuming that the task or job name is *DEMO*, the filename generated will be *demo.sgd*.

All the pertinent information of the nested results will be presented in "partitions blocks". The layout below shows 3 distinct Parts (A,B & C) with 4 partitions blocks in total.



The format is as follows:

```

1  #
2  # AutoNEST V9
3  # SGD File Name = demo.sgd
4  # Layout from NestPro.EXE
5  # ...
6  #
7  JOB = DEMO
8  No of Distinct Shapes = 3
9  Total No of Shapes = 37
10 Total No of Stock Sheet = 1 0
11 Encl Rect = (0.000000 0.000000) (42.854337 50.858266)
12 Stock Sheet = (60.000000 48.000000) x 1
13 Sum of Area of Shapes = 1543.50
14 (A 0.000000 0.000000 0.000000 200.000000 150.000000 2 2 1 0 0.000000 0.000000 0.000000
    0.000000)
15 (B 0.000000 300.000000 0.000000 200.000000 100.000000 3 2 1 0 0.000000 0.000000
    0.000000 0.000000)
16 (C 400.000000 0.000000 90.000000 100.000000 125.000000 4 6 1 0 0.000000 0.000000
    0.000000 0.000000)
17 (B 400.000000 500.000000 0.000000 200.000000 100.000000 1 3 1 0 0.000000 0.000000
    0.000000 0.000000)

```

(Note: The above line numbers are strictly for referencing purposes, they do not appear in the file.)

Line 1 to 6	Lines start with "#" character denote comments. There is no limit to the number of comment lines. The ' AutoNEST V9 ' must be in one of the comment lines. The V9 reference number is used to check the different formats of (.SGD) files for different software releases.
Line 7	SGD filename. The key character here is the "=" equal sign. The words before it are purely descriptive but the name after the sign is important.

Line 8	Number of distinct parts. The key character here is the "=" equal sign. The words before it are purely descriptive but the value after the sign is important.														
Line 9	Total quantity of parts nested. The key character here is the "=" equal sign. The words before it are purely descriptive but the value after the sign is important.														
Line 10	Total number of distinct nested layouts. The key character is the "=" equal sign. The words before it are purely descriptive but 2 values (separated by space) indicate the no. of distinct nested layouts for regular and irregular stocks respectively.														
Line 11-17	The first nested layout information.														
Line 11	Enclosing rectangle of the first nested layout. The first pair of real numbers is the left-bottom point of the rectangle, the second pair is the length and width of the rectangle.														
Line 12	Stock sheet size and the number of the repeated layout. Within brackets are the width and length of the stock.														
Line 13	Sum area of parts nested in the current layout.														
Line 14-17	<p>A list of nested parts. Each line has the following format: (Part-name X Y Angle X_dist Y_dist Y_num X_num Grid_no Leadin_type Leadin_x1 Leadin_y1 Leadin_x2 Leadin_y2)</p> <table> <tr> <td>Part-name</td><td>Name of the part.</td></tr> <tr> <td>(X, Y, Angle)</td><td>Transformation of the part, relative to the first layout's left bottom point.</td></tr> <tr> <td>(X-dist, Y-dist)</td><td>The X and y grid spacing within a partition block.</td></tr> <tr> <td>(Y_num, X_num)</td><td>The number of rows and columns of grid, within a partition block.</td></tr> <tr> <td>Grid_no</td><td>Set to "1" by default. This flag holds the GRID_NO.</td></tr> <tr> <td>Leadin_type</td><td>Set to "0" by default as it is currently not in use. Common Grid Leadin type (a number from 3 to 13) for Common Cutting parts. For non-common cutting parts, it is 0, which means using part individual leadins, corner loops and microjoints</td></tr> <tr> <td>(Leadin_x1, Leadin_y1)</td><td>Set to "0" by default as it is currently not in use.</td></tr> </table>	Part-name	Name of the part.	(X, Y, Angle)	Transformation of the part, relative to the first layout's left bottom point.	(X-dist, Y-dist)	The X and y grid spacing within a partition block.	(Y_num, X_num)	The number of rows and columns of grid, within a partition block.	Grid_no	Set to "1" by default. This flag holds the GRID_NO.	Leadin_type	Set to "0" by default as it is currently not in use. Common Grid Leadin type (a number from 3 to 13) for Common Cutting parts. For non-common cutting parts, it is 0, which means using part individual leadins, corner loops and microjoints	(Leadin_x1, Leadin_y1)	Set to "0" by default as it is currently not in use.
Part-name	Name of the part.														
(X, Y, Angle)	Transformation of the part, relative to the first layout's left bottom point.														
(X-dist, Y-dist)	The X and y grid spacing within a partition block.														
(Y_num, X_num)	The number of rows and columns of grid, within a partition block.														
Grid_no	Set to "1" by default. This flag holds the GRID_NO.														
Leadin_type	Set to "0" by default as it is currently not in use. Common Grid Leadin type (a number from 3 to 13) for Common Cutting parts. For non-common cutting parts, it is 0, which means using part individual leadins, corner loops and microjoints														
(Leadin_x1, Leadin_y1)	Set to "0" by default as it is currently not in use.														

		The starting point of Common Grid Leadin (left bottom corner of leadin rectangle area)
	(Leadin_x2, Leadin_y2)	Set to "0" by default as it is currently not in use. The ending point of Common Grid Leadin (right top corner of leadin rectangle area)

.XML File Format

Altogether there are 3 XML (Extensible Markup Language) files created. They are:-

- Taskname_job.xml
- Taskname_sum.xml
- Taskname_sym.xml

"Taskname_job.xml" is produced when a Task (.job) is being created. The other 2 XML files are being created after a successful nesting, together with .SUM & .SYM files. XML files are useful for exchanging of data between systems and applications over the Internet.

TASKNAME_JOB.XML

```
<!-- AutoNEST V9 ENGLISH -->
<?xml version="1.0" encoding="gb2312" standalone="yes"?>
<!-- Job Name = "BIGPART-2"-->
<!-- Line1: No of Distinct Shapes -->
<!-- Line2: No of Distinct R-Stock & I-Stock & Multi-Stock Control-->
<!-- Line3: Cutting Gap -->
<!-- Line4: Shear Blade Limit -->
<!-- Line5: Left , Right , Bottom and Top side Edge Allowance of the Regular Stock -->
<!-- Line6: Nesting Control Parameters -->
<Job_Contents>
<Distinct_Part_Num> "4" </Distinct_Part_Num>
<Stock Distinct_Rstock_Num = "1" Distinct_Irstock_Num = "0" Multi_Stock_Control = "0"/>
<Cutting_Gap> "15.0000" </Cutting_Gap>
<Shear_Blade_Limit> "0.0000" </Shear_Blade_Limit>
<Edge_Allowance Left = "15.0000" Right = "15.0000" Bottom = "15.0000" Top = "15.0000"
Ir_Stock_Edge_Allowance = "0.0000"/>
<Nesting_Controls Nest1_Type = "D" Stock_Priority = "R" Start_Point = "1" Packing_Dir = "3" Com_Line = "0"
Nest1_Mirror = "0" Ignore_Hole = "1" Save_Remnant = "0" Rem_X_Dim = "0.0000" Rem_Y_Dim = "0.0000"
Max_Stock_Priority = "0" Min_Nest_Space = "0.0000"/>
<Stock Size = "3048.0000 x 9144.0000" Qty = "5" Priority = "1" Cost = "0.00" />
<Part Basic_Qty = "1" Name = "BW1" Orientation = "(ALL)" Pairing = "Y" Priority = "1" Block_Name = "BW1"
Filler_Qty = "0" Support_180 = "1" Step_Angle = "5" />
<Part Basic_Qty = "1" Name = "BW5" Orientation = "(ALL)" Pairing = "Y" Priority = "1" Block_Name = "BW5"
Filler_Qty = "0" Support_180 = "1" Step_Angle = "5" />
<Part Basic_Qty = "1" Name = "BW7" Orientation = "(ALL)" Pairing = "Y" Priority = "1" Block_Name = "BW7"
Filler_Qty = "0" Support_180 = "1" Step_Angle = "5" />
<Part Basic_Qty = "1" Name = "BW8" Orientation = "(ALL)" Pairing = "Y" Priority = "1" Block_Name = "BW8"
Filler_Qty = "0" Support_180 = "1" Step_Angle = "5" />
<Bridge Values = "BRIDGE" Bridge_Width = "20.0000" Corner_Offset = "70.0000" Min_Angle = "50.0000"
Max_Bridges_Per_Part = "4"/>
</Job_Contents>
```

TASKNAME_SUM.XML

```

<!-- AutoNEST V9 ENGLISH -->
<?xml version="1.0" encoding="gb2312" standalone="yes"?>
<Nesting_Summary_Report Task_Name = "BIGPART-2">
<Nesting_Report>
<Stock ID = "#1" Nested_Qty = "1" Size = "3048.00x9144.00" Area=" 27870912.00"
EnclRect_Size="2600.00x8900.61" EnclRect_Area="23141586.00" EnclRect_O_Stock_Area="83.03%" >
<Part Name = "BW1" Nested_Qty = "1" Basic_Qty = "1"/>
<Part Name = "BW5" Nested_Qty = "0" Basic_Qty = "1"/>
<Part Name = "BW7" Nested_Qty = "0" Basic_Qty = "1"/>
<Part Name = "BW8" Nested_Qty = "1" Basic_Qty = "1"/>
<Total_Part_Area>"15574933.06"</Total_Part_Area><Part_Area_O_EnclRect>"67.30%"</Part_Area_O_EnclR
ect>
<Part_Area_O_Stock>"55.88%"
</Part_Area_O_Stock><Total_Part_Perimeter>"28374.83"</Total_Part_Perimeter><Sub_Total>"2/4"</Sub_Tot
al>
</Stock>
<Stock ID = "#2" Nested_Qty = "1" Size = "3048.00x9144.00" Area=" 27870912.00"
EnclRect_Size="3018.00x4153.21" EnclRect_Area="12534387.78" EnclRect_O_Stock_Area="44.97%" >
<Part Name = "BW1" Nested_Qty = "0" Basic_Qty = "1"/>
<Part Name = "BW5" Nested_Qty = "1" Basic_Qty = "1"/>
<Part Name = "BW7" Nested_Qty = "1" Basic_Qty = "1"/>
<Part Name = "BW8" Nested_Qty = "0" Basic_Qty = "1"/>
<Total_Part_Area>"8230940.33"</Total_Part_Area><Part_Area_O_EnclRect>"65.67%"</Part_Area_O_EnclRe
ct>
<Part_Area_O_Stock>"29.53%"
</Part_Area_O_Stock><Total_Part_Perimeter>"22010.78"</Total_Part_Perimeter><Sub_Total>"2/4"</Sub_Tot
al>
</Stock>
</Nesting_Report>
<Nesting_Summary>
<Parts>
<Part Name = "BW1" Basic_Qty = "1" Nested_Basic_Qty = "1" Filler_Qty = "0" Nested_Filler_Qty= "0" Area =
"11458847.66"/>
<Part Name = "BW5" Basic_Qty = "1" Nested_Basic_Qty = "1" Filler_Qty = "0" Nested_Filler_Qty= "0" Area =
"4117914.30"/>
<Part Name = "BW7" Basic_Qty = "1" Nested_Basic_Qty = "1" Filler_Qty = "0" Nested_Filler_Qty= "0" Area =
"4113026.03"/>
<Part Name = "BW8" Basic_Qty = "1" Nested_Basic_Qty = "1" Filler_Qty = "0" Nested_Filler_Qty= "0" Area =
"4116085.41"/>
<Total_Nested_Basic_Qty>"4/4"</Total_Nested_Basic_Qty>
</Parts>
<Stocks>
<Stock Size="3048.00x9144.00" Reg_Stock_Used="2" Used_Stock_Cost ="0.00"/>
<Total Total_Num = "2" Total_Cost = "0.00" />
</Stocks>
</Nesting_Summary>
</Nesting_Reporty_And_Nesting_Summary>

```

TASKNAME_SYM.XML

```

<!-- AutoNEST V9 ENGLISH -->
<?xml version="1.0" encoding="gb2312" standalone="yes"?>
<!-- Process Time Taken -->

```

```

<!-- Input : 0 min 0.01 sec-->
<!-- Pair I : 0 min 0.89 sec-->
<!-- Pair II : 0 min 0.04 sec-->
<!-- Packing : 0 min 0.26 sec-->
<!-- Total : 0 min 1.21 sec-->
<Sym_File_Contents>
<Sym_File_Name>"BIGPART-2"</Sym_File_Name>
<Distinct_Part_Num>"4"</Distinct_Part_Num>
<Total_Parts_Nested>"4"</Total_Parts_Nested>
<Stock Total_Reg_Stock_Used = "2" Total_Irreg_Stock_Used = "0" />
<Stock ID = "#1" Nested_Qty = "1" Size = "3048.00x9144.00" EnclRect_LB_Pt = "(15.000000 15.000000)"
EnclRect_Length = "8900.606602" EnclRect_Width = "2600.000000">
<Total_Part_Area>"15574933.063671"</Total_Part_Area>
<Part Name = "BW1" X_Coord = "793.405704" Y_Coord = "1523.716973" Angle = "0.000000" Display_Color =
"2" Hole_No = "0" Layer = "1" />
<Part Name = "BW8" X_Coord = "6893.683538" Y_Coord = "607.457250" Angle = "0.000000" Display_Color =
"5" Hole_No = "0" Layer = "1" />
</Stock>
<Stock ID = "#2" Nested_Qty = "1" Size = "3048.00x9144.00" EnclRect_LB_Pt = "(15.000000 15.000000)"
EnclRect_Length = "4153.210469" EnclRect_Width = "3018.000000">
<Total_Part_Area>"8230940.327984"</Total_Part_Area>
<Part Name = "BW5" X_Coord = "2011.738470" Y_Coord = "2510.045662" Angle = "0.000000" Display_Color =
"3" Hole_No = "0" Layer = "1" />
<Part Name = "BW7" X_Coord = "2072.339690" Y_Coord = "537.954338" Angle = "180.000000" Display_Color =
"4" Hole_No = "0" Layer = "1" />
</Stock>
</Sym_File_Contents>

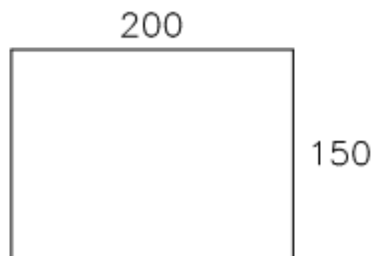
```

.DXF File Format

DXF file is a popular file interchange format among PC-CAD systems. Its exact structure originates from Autodesk, Inc. for the AutoCAD product. Any detailed technical reference should be made to this source.

Example:

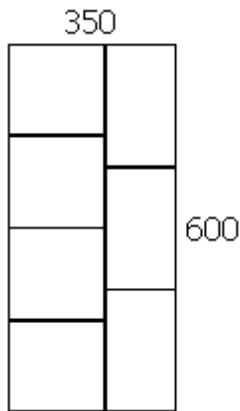
- The following example is a **.DXF** file of a rectangle of 200 x 150, horizontally orientated with its lower left corner at the 0,0 origin.



Column A	Column B	Column C
0	10	21
SECTION	200.0	150.0
2	20	31
ENTITIES	150.0	0.0

0	30	0
LINE	0.0	LINE
8	11	8
0	0.0	0
10	21	10
0.0	150.0	0.0
20	31	20
150.0	0.0	0.0
30	0	30
0.0	LINE	0.0
11	8	11
0.0	0	200.0
21	10	21
0.0	200.0	0.0
31	20	31
0.0	0.0	0.0
0	30	0
LINE	0.0	ENDSEC
8	11	0
0	200.0	EOF

- b. The next example is the same rectangle nested 7 times on a stock sheet of 600 x 350 (Y x X) units, zero cutting gap with the bottom left of the stock sheet inserted at 100, 100 location.



Column A	Column B	Column C
0	10	10
SECTION	450.0	100.0
2	20	20
ENTITIES	100.0	100.0
0	30	30
POLYLINE	0.0	0.0
8	0	0
TASK1	VERTEX	SEQEND
66	8	8
1	TASK1	TASK1
10	10	0
0.0	450.0	LINE

20	20	8
0.0	700.0	0
30	30	62
0.0	0.0	2
0	0	10
VERTEX	VERTEX	300.0
8	8	20
TASK1	TASK1	700.0
10	10	30
100.0	100.0	0.0
20	20	11
100.0	700.0	300.0
30	30	21
0.0	0.0	500.0
0	0	31
VERTEX	VERTEX	0.0
8	8	0
TASK1	TASK1	LINE

Column D	Column E	Column F
8	0	0.0
0	LINE	LINE
62	8	8
2	0	0
10	62	62
300.0	2	2
20	10	10
500.0	300.0	450.0
30	20	20
0.0	500.0	500.0
11	30	30
450.0	0.0	0.0
21	11	11
500.0	300.0	300.0
31	21	21
0.0	300.0	500.0
0	31	31
LINE	0.0	0.0
8	0	0
0	LINE	LINE
62	80	8
2	62	0
10	2	62
450.0	10	2
20	300.0	10
500.0	20	300.0
30	300.0	20
0.0	30	300.0
11	0.0	30
450.0	11	0.0
21	450.0	11
700.0	21	300.0

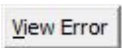
31	300.0	21
0.0	31	100.0
0	0.0	31
LINE	0	0.0
8	LINE	0
0	8	LINE
62	0	8
2	62	0
10	2	62
450.0	10	2
20	450.0	10
700.0	20	300.0
30	300.0	20
0.0	30	100.0
11	0.0	30
300.0	11	0.0
21	450.0	11
700.0	21	450.0
31	500.0	21
1.0	0	100.0
	31	

Column G	Column H	Column I
31	550.0	250.0
0.0	30	31
0	0.0	0.0
LINE	11	0
8	300.0	LINE
0	62	8
62	2	0
2	10	62
10	100.0	2
450.0	20	10
20	400.0	300.0
100.0	30	20
30	0.0	100.0
0.0	11	30
11	100.0	0.0
450.0	21	11
21	250.0	300.0
300.0	31	21
31	0.0	250.0
0.0	0	31
0	LINE	0.0
LINE	8	0
8	0	LINE
0	62	8
62	2	0
2	10	62
10	100.0	2
450.0	20	10
20	250.0	100.0
300.0	30	20

30	0.0	100.0
0.0	11	30
11	100.0	0.0
300.0	21	11
21	100.0	300.0
300.0	31	21
31	0.0	100.0
0.0	0	31
0	LINE	0.0
LINE	8	0
8	0	ENDSEC
0	62	0
62	2	EOF
2	10	
LINE	300.0	
8	20	
0	250.0	
62	30	
2	0.0	
10	11	
100.0	100.0	
20	21	

Error Messages

There are a number of things that can wrong during the running of *AutoNEST*, from corrupted file, wrong software versions, non-conformance of file formats, etc. Generally, these errors will be trapped by the

program through enabling  button. Click the button to display the error message. A disabled View Error button simply means no errors found during processing.

If there are errors, internally, error messages will be stored in a ASCII file called **ANEST.ERR**. ANEST.ERR contains the name of the executing task (e.g. *DXF2VEC*, *Nesting* ..etc.), followed by a brief description of the error and the actions to be taken to correct them.

Categories of Errors

We have grouped errors into the following six categories with corresponding numbering procedure.

	Error Number	Category
a.	100-199	I/O Error
b.	200-299	Limits Conformance Error
c.	300-399	File Format Conformance Error
d.	400-499	Warning Message
e.	500-599	System Error
f.	600-699	Fatal Error

The detailed explanation of each possible error is in an ASCII file called **ERROR.MSG**. It is structured with a prefix each error number as follows:

#E Error number with error message

- #D Error number with description
 #S Error number with proposed solution to overcome problem

The contents of the **ERROR.MSG** are given as below.

```
# AutoNEST English Version
# File Name: error.msg /* AutoNEST V9 */
# Date: Dec. 2002
# 100-199: I/O Error
# 200-299: Limits Conformance Error
# 300-399: File Format Conformance Error
# 400-499: Warning Message
# 500-599: System Error
# 600-699: Fatal Error
0 Press any key to EXIT
```

Reference To Error Messages

- 100E *The hardware lock has been disconnected, removed, faulty or cannot be detected.*
 100S *1. If hardware lock has been removed or loosened, please attach the hardware lock to the computer's parallel port connector.*
 100S *2. Install the hardware lock driver from the CD-Rom, if you have not done so.*
 100S *3. Should the hardware lock be damaged, please report to your dealer.*
- 101E *Authorization File is incompatible or corrupted.*
 101S *1. Please copy the original Authorization File from the Install CD-Rom/Disk to the main directory where the program is installed.*
 101S *2. If the Authorization File is incompatible, please contact your dealer.*
- 103E -
 103E *Cannot support unlimited ('M') part qty.*
 103S *Change the unlimited part qty to be limited.*
- 104E -
 104E *Cannot support coiled stocks.*
 104S *Change the coiled stock to be normal regular stock.*
- 105E -
 105E *Cannot support unlimited ('M') part qty for tasks with multiple distinct parts.*
 105S *Change the unlimited part qty to be limited.*
- 106E -
 106E *Cannot support coiled stocks for tasks with multiple distinct parts.*
 106S *Change the coiled stock to be normal regular stock.*

- 107E *Number of distinct stock sizes cannot be ≤ 0 .*
107S *Specify a num (≥ 1).*
- 108E *Number of distinct parts is greater than the actual*
108E *number of parts in the task file.*
108E *Add more parts in task file.*
- 109E *Number of distinct parts cannot be ≤ 0 .*
109S *Specify a num (≥ 1).*
- 110E
110E *Invalid number of parameters.*
110E
110E *Format : program_name task_name*
110E
110E *Example : NESTPRO task 1*
- 112E *For "UpdateNest", the part output format should be*
112E *set to "DWG".*
112S *Run "SYSDATA" to re-set the "File Format" to "DWG".*
- 114E *Coiled stock and unlimited ('M') part qty*
114E *is not supported.*
114S *1. Either change the stock definition.*
114S *2. Or change part qty "M" to a fixed qty.*
- 115E *Assembly/Cluster needs more qty for the following part -*
115S *Please re-define the part qty.*
- 130E *No .DXF files found in directory -*
130D *The AutoNEST data files cannot be found in default*
130D *directory. Check part directory in <SYSDATA>.*
- 131E *No .VEC files found in directory -*
131D *The AutoNEST data files cannot be found in default*
131D *directory. Check part directory in <SYSDATA>.*
- 133E *No .SYM files found in directory -*
133D *The AutoNEST data files cannot be found in default*
133D *directory. Check task directory in <SYSDATA>.*
- 134E *No .DWG files found in directory -*
134D *The AutoNEST data files cannot be found in default*
134D *directory. Check part directory in <SYSDATA>.*
134D *Only applicable for AutoCAD.*
- 136E *No .STK files found in directory -*
136D *The AutoNEST data files cannot be found in default*
136D *directory. Check part directory in <SYSDATA>*
- 140E *Can't open file for read -*
140D *The error will happen when AutoNEST cannot open*
140D *the data file for read.*

- 140D *Eg, In <Nesting>, please make sure that the task file*
140D *and the part files are in the specified default*
140D *directories.*
- 141E *Can't open AutoNEST file for read -*
141D *The AutoNEST supporting files are missing or corrupted.*
141S *Check the AutoNEST distribution diskettes, and re-load*
141S *the necessary files onto the hard-disk.*
- 142E *Can't open file for read -*
142D *Need to create the above data file.*
- 143E *Authorization file not found.*
143S *Please consult your dealer.*
- 144E *All concurrent user licenses are currently being used.*
144S *Please wait a while until one of the licenses has*
144S *been released.*
144S *You can also contact your dealer to increase the*
144S *network concurrent user licenses.*
- 145E *Your current license is not for this product.*
145S *Please contact your dealer.*
- 146E *Trial license has expired.*
146S *Please contact your dealer.*
- 147E *Authorization License Code not found.*
147S *Submit the "Locking Code" and the*
147S *"License Code Request Form" duly filled-in to your dealer.*
- 148E *The system date has been tampered with.*
148S *Please contact your dealer.*
- 149E *Can't open file for read -*
149D *The error will happen when AutoNEST cannot open*
149D *The data file for read.*
149S *1. The Task must be NESTED first before Auto-Bridge can be*
149S *performed.*
149S *2. Make sure that the Part & Task directories are correctly*
149S *specified in the SYSDATA.*
- 150E *Can't open file for write -*
150D *The error will happen when AutoNEST cannot open*
150D *the data file or supporting file for write.*
150S *1. Run Out of File Handle, increase the "FILE" field in*
150S *config.sys.*
150S *Eg FILES=20*
150S *2. Hard Disk Full.*
- 151E *Can't open file for append*
151D *The error will happen when AutoNEST cannot open*
151D *the data file or supporting file for write.*

- 151S 1. Run Out of File Handle, increase the "FILE" field in
151S config.sys.
151S Eg FILES=20
151S 2. Hard Disk Full.
- 153E The AutoNEST file is corrupted -
153E Check the AutoNEST distribution diskettes, and re-load
153E the necessary files onto the hard-disk.
- 154E The file is corrupted -
154S Report to AutoNEST dealer
- 155E The following part(s) cannot be found in the job file.
- 156E No part found in the job file.
- 157E No part selected.
- 158E No stock selected.
- 159E Cannot find job file, therefore ViewNEST terminates here.
- 160E "Pre-defined" parts cannot be read.
160D This is a trial version, only "pre-defined" parts
160D in the default "...\\parts" directory can be nested.
160S Change the "Part/Ir-Stock Directory" to the
160S installation default "...\\parts" directory and try again.
- 201E No suitable stock size at all
201D During nesting if all the declared stock sizes are
201D found to be too small to nest even one or a pair of the
201D smallest part in the task, this message will be displayed.
201S Either increase the stock sheet size or nest smaller parts.
- 202E Number of distinct parts exceeds 1000.
202D The number of distinct parts submitted in the task file
202D has exceeded the limit of 1000.
- 203E Number of distinct stocks exceeds 500.
203D The number of distinct stocks submitted in the task file
203D has exceeded the limit of 500.
- 204E Part profile has more than 1500 vertices -
204D The number of vertices in one of the profile of the part
204D has exceeded the maximum number allowed : 1500.
- 205E No nesting layout meets minimum utilization ratio requirement.
205D All nesting layouts are abandoned.
205S Either change stock size or set a lower minimum utilization ratio
- 206E Ir-stock profile is out of processing limit.
206D This may be due to too many vertices in the ir-stock
206D profile or too many holes defined inside the ir-stock.

- 206S *Please simplify the ir-stock profile or try to reduce*
206S *the no of holes inside the ir-stock.*
- 300E *Incompatible AutoNEST version -*
300D *Error happens when data file is from a different version*
300D *of AutoNEST. For eg, the data file for AutoNEST V9*
300D *will contain the following comment line :*
300D *"# AutoNEST V9"*
300D *The Release no is used to check the compatibility of*
300D *different formats of data files between different releases.*
300D *AutoNEST provides downward compatibility for all the*
300D *data files.*
300S *Please make sure all the files are not corrupted and*
300S *the Release No is properly designated.*
300S *If the file is corrupted, re-create the file.*
- 305E *Profile(s) not closed or contains irregularities at the following location(s) :*
305S *Either re-create the part/ir-stock or change the aperture*
305S *value in ANEST.TOL.*
- 306E *Profile(s) not closed or contains irregularities -*
306S *Either re-create the part/ir-stock or change the aperture*
306S *value in ANEST.TOL.*
- 307E *Profile has lines that CROSSES over -*
307D *The profile has some intersection lines.*
307S *Either re-create the part or change the aperture value*
307S *in ANEST.TOL.*
- 308E *Profile has multiple SAME point -*
308S *Either re-create the drawing or change the aperture value*
308S *in ANEST.TOL.*
- 312E *Profile is an ACAD Block - need to EXPLODE*
312D *Only applicable for AutoCAD*
312S *Re-create the drawing.*
- 313E *Profile contains SPLINE segment -*
313D *AutoNEST cannot accept profile with "SPLINE" segment*
- 314E *Cannot convert DXF file -*
314D *No profile can be detected on the LAYER or of the COLOR*
314D *as specified in the SYSDATA "Input Part/Ir-Stock*
314D *- Layer Setting".*
- 315E *Part has TOO MANY EDGES -*
315D *The constraints for vector compression is too tight.*
315S *Change the constraints in ANEST.SYS.*
315S *Either decrease minimum vector, increase total area, or*
315S *increase relative global tolerance.*
- 340E *Invalid file format for -*
340S *Need to re-do*

- 341E *Invalid file format for AutoNEST supporting file -*
341S *Need to re-load from Diskette*
- 345E *Cannot find "ANEST" layer in .DXF file -*
345S *Need to save blocks in "ANEST" layer.*
- 350E *Cannot find BLOCK in .DXF file -*
350S *Need to re-create file*
- 351E *Cannot find ENTITIES in .DXF file -*
351S *Need to re-create file*
- 352E *The .DXF file is above R12 version -*
352D *Current software cannot support part/ir-stock*
352D *DXF file above R12 version in Nest Manager environment.*
352S *Save your part/ir-stock .DXF file as in R12 format.*
- 400E *Not all parts are nested. It is due to either insufficient stock or*
400E *no suitable stock size found.*
400E *For more information, look at the summary.*
- 401E *Pattern type of the following boundary data is wrong -*
- 402E *Some parts are not nested as nested layout is below*
402E *Utilization Factor.*
402E *For more information, look at summary / schedule.*
- 403E *Profile(s) not closed or contains irregularities at the following location(s) :*
403D *This profile(s) will be ignored during nesting.*
403D *Please check the profile(s) carefully, especially for*
403D *outer-profile. Then save again to ensure correct nesting result.*
- 404E *Current program can only support 0,90,180 degrees for stock*
404E *grain direction. Other angles are ignored during nesting.*
- 405E *The following changed or missing part(s) are recovered back:*
405D *This is a trial version, only "pre-defined" parts can be nested.*
405D *The "pre-defined" parts are not allowed to be modified.*
- 406E *The following custom part(s) cannot be nested:*
406D *This is a trial version, only "pre-defined" parts can be nested.*
406S *Delete the custom part(s) from the Task and nest again.*
- 500E *Not Enough Memory.*
500D *Error due to insufficient memory to execute the AutoNEST program.*
500S *Remove all unnecessary memory-resident programs.*
500S *If in AutoCAD environment, check acad.pgp for memory reserve*
500S *for all the programs. If possible, increase the memory reserve*
500S *for the program. [For more information, please refer to AutoCAD*
500S *Reference Manual, Appendix B]*
- 600E *Pointer out of range*

600D *Software bugs.*
 600S *Report to AutoNEST Dealer.*

601E *Array Out Of Range*
 601D *Software bugs.*
 601S *Report to AutoNEST Dealer.*

602E *Part expansion error -*
 602E *Expanded part contains invalid geometry*
 602D *The part after expansion to the specified cutting gap*
 602D *may contain invalid geometry or the number of vertices*
 602D *of the expanded part exceeds the limit : 350 vertices.*
 602S *Re-create the part by reducing the number of vertices.*
 602S *This may help to solve the problem. If not, Report it to*
 602S *AutoNEST Dealer.*

652E *Nesting Error: -*
 652E *Report to AutoNEST Dealer*

System Parameters

In AutoNEST, there are three system files that affect and control the performance of NestPRO, the nesting engine and other functions. They are :

- ANEST.SYS** ANEST.SYS contains all the relevant and essential variables that affect parts approximation in **Nesting**.
- ANEST.TOL** ANEST.TOL contains the '**aperture**' value for **SavePart / MirrorPart** and part expansion control for cutting gap consideration and arc approximation for nesting speed and quality.
- ANEST.SET** ANEST.SET contains parameters that will interface with non-AutoCAD systems, e.g. AutoSketch, Generic CAD, where a full **DXF** is required or the layer names are restricted to numeric.
- ANEST.SET also contains the default values for **TaskEdit**, default display distances between nested layouts and Auto-Bridge.

Format for ANEST.SYS

```
#
# AutoNEST V9
# File Name= ANEST.SYS
#
@SHP_APPROX
25      # Minimum vectors
5        # Maximum No of loops
25      # Minimum concave area increment (in percentage)
8.0     # Total Area Increment (in percentage)
2        # Minimum Arc (in percentage)
0.08    # Global tolerance factor (wrt largest enclosing rectangle edge)
3.0     # Local tolerance factor (wrt neighbouring edge)
0.0     # Circle & arc accuracy degree, range from 0.0 to 9.0
100     # Minimum vectors per spline
```


@NEST_CONTROL

3 # 0: No Nestb 1: Normal + Nestb 2: Nestb Only 3: As indicated in UI
 30 # Nestb maximum group number
 1 # Nestb minimum average number per group
 0.1 # Nestb flexible ratio (0.0 to 0.1)
 0 # 1: Nestb keep grid, 0: break grid
 0 # Allow / Disallow high priority parts nested in the hole of low priority parts
 0 # Nest rectangle parts in regular grids (only when all parts are rectangle)
 2 # 0: Stock priority- system decides based on best utilization; 1: Follow stock priority strictly;
 2: Follow stock priority strictly but allow merging of stocks wherever possible.

 0 # 1: Follow part priority strictly; 0: Part priority - system decides based on best utilization
 500 500 # Maximum number of distinct stock sizes, ir-stock sizes
 0 # Maximum number of distinct parts (in the future)
 60 40 # Stocks Merging Control: merge only when utilization is < N1%(rectangles) or N2%(non rectangle)

@MORE_NEST_CONTROL

400 400 # Best Yield Option: Maximum no. of distinct Parts and Stocks
 200 # No. of pins per row, min.=100, max.=500 (higher value may give better result but will slow down nesting)
 1 # SPCN - 1: Whole part must be inside Small-Part Zone; 2: CG must be inside Small-Part Zone
 0 # SPCN - The percentage of extension allowed for Small-Part zone (eg 10)

@REMNANT_CONTROL

1 # 0 : Fill whole stock with filler parts.
 # 1 : Fill stock with filler parts up to the basic qty edge)
 0.10 # If last stock (after nest the basic qty parts) remains 10% or less,
 # fill up the rest of stock with filler parts.
 0 # 1 : Rectangularisation of remnants, 0: No
 1 # 1 : Retain profile of original Stock; 0:No
 remnant # Suffix for Remnant Stock (Ir-Stock)
 IR-remnant # Suffix for Remnant of Ir-stock
 0.5 # Gap between Part & Remnant (factor of cutting gap)

@SIMPLIFY_CONTROL

200 # minimum number of vertice of polygon to do simplification in PIP
 # adjust higher will slow down nest but will get better PIP pattern
 0.05 # Part simplification ratio to remove concave gaps (0.00 - 0.10)

@BRIDGE_CONTROL

140 # Maximum angle between vectors that is valid for Corner Offset consideration
 0 # 0: Draw bridge as per normal. 1: Draw bridge with overlap
 0.97 # To draw commonline (toolpath) using Part Enclosing Rect. Ratio of Part Area
 (against Enclosing Rect) must be this value or more.
 0 # To draw commonline (toolpath) using Part Enclosing Rect. 0.6 (or specified) of Length & Width must
 be orthogonal
 0.001 # To draw commonline (toolpath). Tolerance Value for Rectangle

@TEXT_DISPLAY_CONTROL

4 # Tolerance angle for Part Labelling
 0.80 # Text Height scale factor (in order to fit text within a Part's width)
 0.95 # Text Height scale factor (in order to fit text within a Part's length)
 0.15 # Auto Text Height – Factor of Text height/Part width

@OTHER_CONTROL

0 # Output Part color code (for debugging) {0: As in Sysdata, 1: Nesting Logic, 2: SPCN}

```

0      # Display stock layouts strictly in packing order (for debugging)
1      # Fill layout space at the end of packing for every stock (last PIP) (for debugging)
0      # Label part by part name-0, by priority-1
0      # 1: Parts & Task directories default to same; 0: Remain separate directories
1      # 1: Excel reports created during Nesting; 0: No Excel reports created during Nesting
0      # 1: Text(Partname, Taskname...) in reports to be in uppercase; 0: Text case to remain as original

```

@SHP_APPROX

Under this heading are the parameters that influence how a polygon profile is being approximated. Generally, the rougher the approximation, less number of vectors will be used to represent the geometry. Fewer vectors will decrease the workload of the nesting process.

The user can judiciously adjust the following parameters for maximum performance during the nesting process.

Profile approximation is therefore more effective when the original number is large. However too much approximation may also lead to profile distortion that provides inefficient nesting.

1st Parameter : Minimum Vector (for Polygon Approximation= 25)

It is the minimum number of vectors in one polygon profile before part approximation routine is allowed. Generally, a polygon profile with less than 25 vectors when approximated would produce a new profile that is quite different from the original part. Therefore a limit is imposed to prevent unexpected results.

2nd Parameter : Maximum No of Loops (= 5)

The maximum number of times the subroutine processes through all the vertices of a profile. It acts as a safety counter to ensure that the subroutine will terminate properly. However it is to be noted that complex profiles require more approximation passes. Too many passes will make the process longer, too little will not give an efficient approximation. A judicious number will be derived from the user's experience.

3rd Parameter : Minimum Concave Area Increment (in percentage= 25)

This parameter has an important role in controlling the amount of "cut-through" in concave portion of the polygon profile. It is the incremental area allowed for all the "cut-through" in concave approximation. Increase in the parameter will result in greater amount of "cut-through" during approximation. Note that another parameter, 'Minimum Arc', does play a similar role in controlling the amount of "cut-through" as this control parameter. 'Minimum Arc' will be discussed later in the section.

4th Parameter : Total Area Increment (in percentage= 8)

As the routine approximates a polygon profile, the net area will be expanded. A limit in area increment is therefore necessary before the new profile is grossly different from its original profile.

This value is the additional percentage in area increment that is allowed for approximation subroutine. The approximation will be terminated once the area increment exceeds the value of parameter or in the case when there is no more approximation possible.

Do not confuse this parameter with the 'Minimum Concave Area Increment' discussed previously. The present parameter is using the total area of profile after approximation in deciding the quitting of subroutine. The previous parameter is used during the concave portion of the profile,

5th Parameter : Minimum Arc

This parameter is not valid for current version.

6th Parameter : Global Tolerance Factor (= 0.08)

7th Parameter : Local Tolerance Factor (= 3.0)

8th Parameter : Circle & Arc Accuracy Degree (= 0.0)

For parts with a very high number of vectors or arcs, this parameter will enable you to control the "roughness". The higher the value, the higher the accuracy of arc vectorization. The default value 0 will give a reasonable result to most jobs. Bear in mind that the higher the value, the more time the nesting will take.

9th Parameter : Minimum vectors per spline (= 100)

This parameter enables you to set the minimum no. of vectors to be created upon converting a SPLINE entity in .DXF to .VEC. The maximum no. of vectors, after conversion of spline is 1,000. This parameter is used in **ConvertPart (DXF2VEC)**.

@NEST_CONTROL

There are 12 parameters under @NEST_CONTROL where the first 5 parameters control the NESTB nesting engine. Each of them are explained below :-

1st Parameter :

- 0 Do not use the exhaustive nesting engine, NESTB
- 1 Use the normal nesting engine OR NESTB depending on the TASK (.job)
- 2 Always use the exhaustive nesting engine, NESTB.
- 3 As indicated in the UI. By default, option 0 is used for nesting. If the "Extended" check-box is marked, then option 1 will be employed.

2nd Parameter :

- 30 Increasing this number will enable more Tasks to be submitted to the NESTB nesting engine. At the same time, less of "similar" shaped parts will try NESTB.

3rd Parameter :

- 1 Default is "1". This is the lower limit of the average number of individual parts in a Task for NESTB.

4th Parameter :

- 0.1 This means that NESTB is able to extend beyond 10% from the nested pattern when considering if a part or a group of parts are suitable.

5th Parameter :

- 0 If there is a grid pattern consisting of identical shapes in the nesting result. NESTB engine will ignore the regularity of the grid, and will carry out any necessary operation to make the final result tighter.

- 1 If there is a grid pattern consisting of identical shapes in the nesting result. NESTB engine will not break the grid, and retain its regularity in the final output.

6th Parameter :

- 0 Do NOT allow high priority parts to be nested into the hole of low priority parts
1 Allow high priority parts to be nested into the hole of low priority parts

7th Parameter :

- 0 Nest rectangle parts using generic nesting engine.
1 Nest rectangle parts in regular grids for better common-line tool-path generation (only when all parts are rectangle) - using RNEST engine.

8th Parameter :

- 0 Stock Priority - Based on the priority specified, *AutoNEST* decides based on best utilization.
1 Follow strictly Stock Priority as defined in .JOB.
2 Follow stock priority strictly but allow merging of stocks when the stock utilization is less than the specified values set in the 12th parameters under @NEST_CONTROL

9th Parameter :

- 0 Part Priority - Based on the priority specified, *AutoNEST* decides based on best utilization.
1 Follow strictly Part Priority as defined in .JOB.

10th Parameter :

- 500 The maximum number of distinct regular stocks defined in a task.
500 The maximum number of distinct irregular stocks defined in a task.

11th Parameter:

- 0 The maximum number of distinct parts defined in a Task (currently NOT in use). Currently this value is fixed at 1000.

12th Parameter:

- 60 Stocks Merging Control : merge the stocks if the utilization is less than 60% - applicable for rectangles.
40 Stocks Merging Control : merge the stocks if the utilization is less than 40% - applicable for non-rectangles.

@ MORE_NEST_CONTROL

There are 2 parameters under @MORE_NEST_CONTROL. They are explained below :-

1st Parameter :

- 400 400 Applicable for “**Best Yield**” and “**Lowest Cost**” options in **Multi-Stock Control**, the maximum no. of distinct Parts and Stocks allowed in any one Task.
The default is 400 for both.

2nd Parameter :

- 200 This parameter is used in the “Special Pins Nesting” routine incorporated into the nesting engine.

No. of pins per row, min.=100, max.=500 (higher value may give better result but will slow down nesting)

3rd Parameter :

- 1 This parameter is used in the Small-Part Nesting Control (SPCN) engine.
1 denotes the Part must be inside the Small-Part zone; 2 denotes the CG of the Part must be inside the Small-Part zone.

4th Parameter :

- 0 This parameter is used in the Small-Part Nesting Control (SPCN) engine.
This parameter is the percentage of extension allowed for Small-Part zone.
0 means no extension. 10 means Small-Part zone is extended by 10%

@REMNANT_CONTROL

There are 7 parameters under @REMNANT_CONTROL. Each of them are explained below :-

1st Parameter :

- 0 Always fill-up the whole stock with filler parts.
1 Fill stock with filler parts, only up the edge of the "Basic qty" parts.

2nd Parameter :

- 0.10 If last stock (after filling in the basic quantity) remains 10% or less, then fill up the rest of the stock with filler parts, otherwise do not.

3rd Parameter :

- 0 Created stock Remnants will NOT be rectangularized or broken up into rectangles.
1 Created stock Remnants will be rectangularized and saved into individual rectangles.

4th Parameter :

- 0 Do not include the original stock profile for the created remnant.
1 Include the original stock profile for the created remnant.

5th Parameter :

- \$REM Suffix for remnant stock name.
The names of the remnant stocks automatically created are :- *Taskname_\$REM1-1.stk*, *Taskname_\$REM2-1.stk*, *Taskname_\$REM2-2.stk* and so on.
The naming convention is as follows :-
*_ \$REM(LAYOUT_NO)-(NO) where * is the taskname and "\$REM" the suffix.

6th Parameter :

- IR-remnant Suffix for remnant of ir-stock
The names of the remnant(s) of ir-stock automatically created are :-
Taskname_IR-remnant1-1.stk, *Taskname_remnant1-2.stk*, *Taskname_IR-remnant2-1.stk*
The naming convention is as follows :-
*_ (LAYOUT_NO)-(NO) where * is the Taskname and "IR-remnant" the suffix.

7th Parameter :

- 0.5 Gap between the remnant and the Parts. (factor of the cutting gap)

@SIMPLIFY_CONTROL

There are 2 parameters under @SIMPLIFY_CONTROL. Each of them are explained below :-

1st Parameter :

200 Minimum number of vertices of polygon to do simplification in PIP, adjusting higher will slow down nesting but will get better PIP pattern.

2nd Parameter :

0.05 Part simplification ratio to remove concave gaps (0.00 to 0.10). Higher value will make nesting fast, but waste more stock material; lower value will use more nesting time, but waste lesser stock material.

@BRIDGE_CONTROL

There are 5 parameters under @BRIDGE_CONTROL that controls Auto-Bridge.

1st Parameter :

140 The maximum angle in degrees formed by 2 vectors that is valid for **Corner Offset** consideration specified by user. If the 2 vectors formed an angle larger than this parameter, then the program will ignore the **Corner Offset**. (range 90~180 deg)

2nd Parameter :

0 Draw bridge as per normal. See Figure A.

1 Draw bridge with overlap. This is to enable the cutting tool path to overlap at the bridge entrance /exit. (See Figure B - follow the sequence of the arrows)

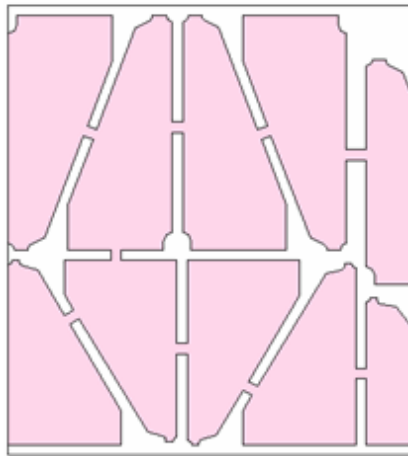


Figure A

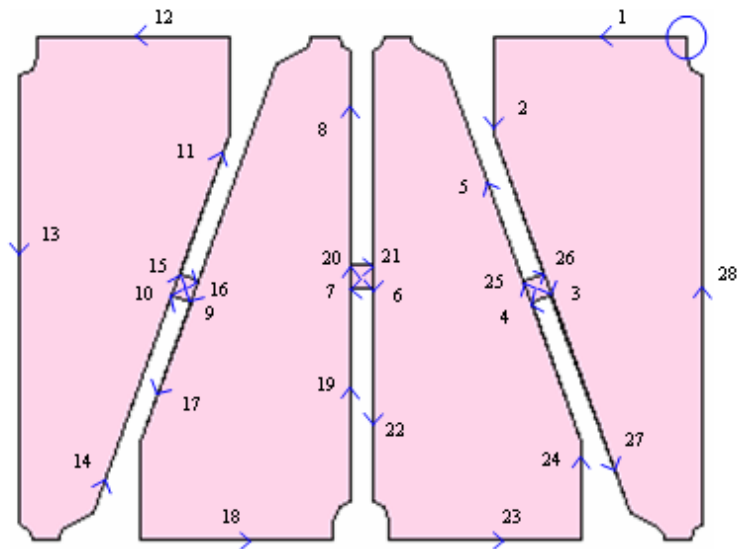


Figure B

3rd Parameter :

0.97 To draw commonline (toolpath) using Part Enclosing Rect. Ratio of Part Area (against Enclosing Rect) must be this value or more. See Figure C.

4th Parameter :

0.6 To draw commonline (toolpath) using Part Enclosing Rect. 0.6 (or specified) of Length & Width of Part must be orthogonal.

5th Parameter :

0.001 To draw commonline (toolpath). Tolerance Value for Rectangle.

This is the tolerance value Commonline will consider when determining if a part is rectangle.

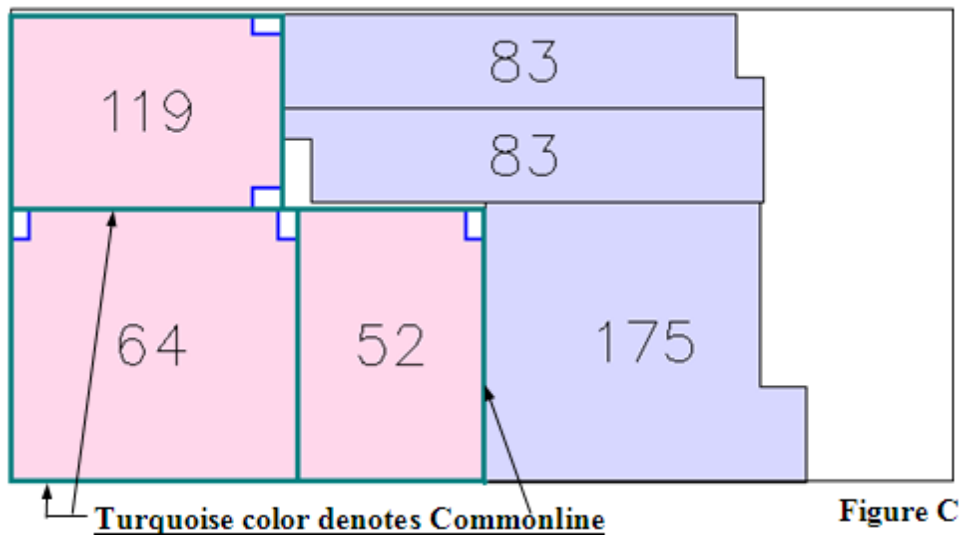


Figure C

Commonline (toolpath) will consider Parts 119, 64 & 52 as rectangles and then add "L" for toekick.

Area of Part/ Area of Enclosing Rect must be 0.97 or greater.

Whereas Parts 83 & 175 are not considered as rectangles because their Area of Part/ Area of Enclosing Rect < 0.97.

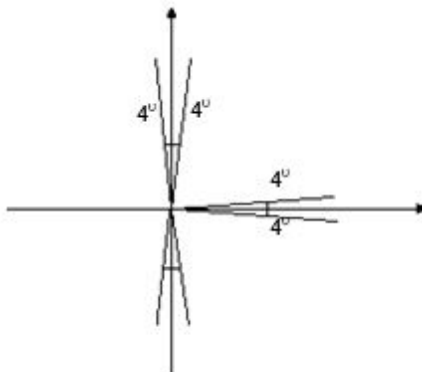
The toolpath for Parts 83 & 175 will be the same as their profile -NO commonline.

@TEXT_DISPLAY_CONTROL

There are 4 parameters under @TEXT_DISPLAY_CONTROL that controls the display of text labeling.

1st Parameter :

- 4 Tolerance angle for Part Labelling. Text orientated within 0 to 4 degrees reference to the X/ Y axis will be snapped to the nearest axis. See below.

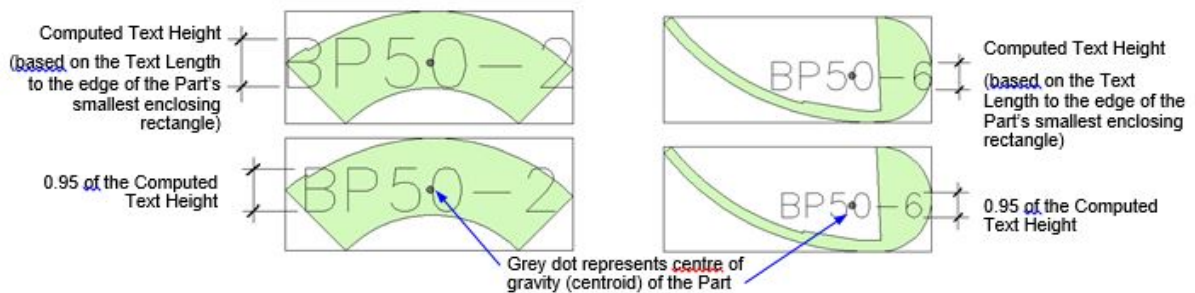
**2nd Parameter :**

- 0.8 Text Height scale factor – applicable if “Reduce Scale to Fit” is set to ON (in order to fit text within a Part's width). Especially for thin narrow parts. See diagram below for illustration.



3rd Parameter :

- 0.95 Text Height scale factor – applicable if “Reduce Scale to Fit” is set to ON (in order to fit text within a Part's length). This scale factor is based on the computed Text Height. See diagram below for illustration.



4th Parameter :

- 0.15 'Auto' Text Height is calculated based on the smaller width of the Part's enclosing rectangle multiply by this factor.

@OTHER_CONTROL

The parameters under this heading are mainly for debugging purposes. Please do not change any one of them. However you can set both the PART and TASK directories to default to the same by changing the 5th parameter to “1”.

Format for ANEST.TOL

```
#
# AutoNEST V9
# Filename = anest.tol
#
# Angle tolerance=Number of vertices(minimum),angular allowance
@ANGTOL
5 2.000000
10 5.000000
20 10.00000
60 20.00000
#
# APERTURE=Size of the aperture
@APERTURE
0.10
#
```



```
# Shape Expansion=(Max) angle to trim, trim ratio
@SHP_EXPAND
90 1.0
#
# Arc APPROXIMATION ERROR TOLLERANCE (>=0.001)
@ARC_APPROX_ERROR_TOLL
0
0.01
@ARC_90_APPROX_NUM
48
#
# Shear Stock Tolerance = absolute value that packing can exceed the stock boundary
@SHEAR_STOCK_TOL
0.1 0.039
#
# Cluster-Part Tolerance = tolerance values for checking of Part rotation
@CLUSTER_PART
0.1 0.03 0.001
#
# SYM File Part Sorting Tolerance
@SYM_SORTING
0.0005
```

@ANGTOL

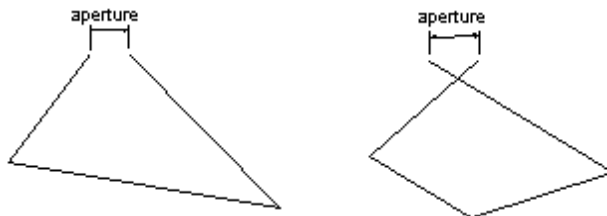
This parameter is not valid for current version.

@APERTURE

Under this header, you can specify a value (floating point) for the 'Aperture' to be used in **SavePart/MirrorPart**. This aperture value allows you to control the degree of accuracy when considering whether two distinct end points actually meet.

If the current units is Metric/millimeters, then the value is taken as millimeter(s). On the other hand, if the units is Imperial/inches, then the value is taken as inch(es).

'Aperture' is the degree of accuracy of 2 very close but distinct points to be considered as the same point. Therefore, polygons with vectors not entirely closed or overlapping will still be tolerated if the distance apart is within the default aperture. See below:



@SHP_EXPAND

Under this header, you can specify 2 parameters to control part expansion for cutting gap consideration. These parameters correspond to the maximum angle between 2 vectors at a vertex and the trimming ratio.

Part expansion occurs at two places: during the execution of **Nesting** or **BatchNesting** and **ExpandPart**.

After a part has been expanded, the expanded profile may consist of long narrow and pointed edges. These edges will cause obstruction during the nesting process. It is therefore advisable to trim off these pointed edges. The amount of trimming required for your application is defined here.

The first parameter indicates the range of angles to be considered for trimming. Default is 90, which means that any angle less than 90 will be trimmed. The second parameter specifies the amount to be trimmed (ratio of cutting gap) at the vertices of the part/shape. Default is 1.0.

@ARC_APPROX_ERROR_TOLL

Under this header, you can specify 2 parameters to control arc or circle approximation if a part has arcs or circles as profiles.

The first parameter can be set to 1/0. 1 means this parameter is on and 0 means off. If it is off (0), the second parameter will be ignored. Otherwise the second parameter means the error tolerance when approximating arcs or circles. The smaller the value is, the less distortion the approximated profile is. For instance, if the first parameter is set to 1 and the second parameter is set to 0.01, after approximating a circle part, the new part radius will increase at most 0.01.

It's recommended that the first parameter be set to off (0), i.e., users do not have to use this control parameter. The system will handle arc approximation automatically.

Only when you have some very big parts which are very close to the stock sizes, you may need to set this parameter on to adjust arc approximation to fit the parts. Otherwise you may not be able to nest those big parts.

@ARC_90_APPROX_NUM

This parameter will determine the number of vectors to simulate a 90-degree arc. The higher the number, the more vectors will be generated. Thus affecting the nesting speed, making it to take longer time. Therefore it is suggested that this parameter be maintained unless in special circumstances. For example, a very big curved part that fit into the Stock exactly.

@SHEAR_STOCK_TOL

There are two values – the first is for metric units and the other for imperial units. This is a tolerance used for packing in Shear nesting.

@CLUSTER_PART

There are three values and all of them are used by **SaveClusterPart**. The first 2 values are tolerances used when comparing the vertices of a rotated part. The first is for metric units and the second for imperial units. The third value is the tolerance for angle of rotation of part.

@SYM_SORTING

This is the tolerance value used during the sorting of Parts for .SYM file.

Format for ANEST.SET

This file is used for interfacing with CAD systems other than AutoCAD. It also contains the default values of **TaskEdit**, default display distances between nested layouts and Auto-Bridge.

A sample **.DXF** file is required. To create **sample.dxf** file, open a drawing called SAMPLE in the particular CAD software used and do a "DXF-OUT" equivalent command.

Format for ANEST.SET

```

#
# AutoNEST V9
# Filename = anest.set
#
# 1. Shape dxf sample filename
# 2. Shape dxf option (1 BLOCK/0 ENTITIES)
# 3. Nested layout dxf sample filename
# 4. Nested layout dxf option (1 BLOCK/0 ENTITIES)
# 5. Insertion point 0: Shape box center; 1:Fix insertion as below; 2:Use DXF "INSBASE" value
# 6. Insertion point x and y
# 7. Layer option (1 to set default layer name)
# 8. Name of default Layer
# 9. Layer name of Enclosing Rectangle of nested layout
# 10. Color of Enclosing Rectangle of nested layout
#
sample
0
sample
0
1
0.00 0.00
1
1
$ENCL_RECT_RESERVED_LAYER$
7

@TASK
$NULL$ # Default Title
$NULL$ # Default Job Ref Name
$NULL$ # Default Project Name
$NULL$ # Default Client
$NULL$ # Default Drawn By
$NULL$ # Default Material Grade
$NULL$ # Default Thickness
$NULL$ # Default Unit Weight
0.00 0.00 # Default Cutting Gap and Shear Blade Limit
0.00 0.00 0.00 0.00 0.00 # Default Edge Allowance (Left, Right, Bottom ,Top) & ir-stock
C R 1 3 0 0 1 0 600.0000 600.0000 0 0.00 # Default Nest Options (format :same as in .JOB)
STOCK = # Default Stock Format : STOCK = Width x Length Qty Priority Cost-per-Stock
IR_STOCK = # Default Ir-Stock Format : IR_STOCK = Name Qty Priority
0 # Default Multi Stock option- 0:Auto; 1: Extended Best Yield; 2: Extended Lowest Cost
1 # Default Part +180 Orientation 1:Support +180; 0:No
5 # Default Nesting Step Angle (Integer, Recommended:5, Range:0 to 360)
ALL # Default Part Orientation (0 90 180 or step 3 0 360 or ALL)
1 # Default Part Pairing: 1: Yes; 0: No
3 1 5 # Default Stock Priority, Ir_stock Priority and Part Priority

WxL # Stock Format for IMPORT STOCKS. Accepted formats: WxL or LxW

```

0 # Top-up if stocks are insufficient : 1: Yes; 0: No

@BRIDGE

0 # View nesting result with bridge
 10.0 0.25 # Default bridge width (metric imperial)
 20.0 0.5 # Default corner offset (metric imperial)
 45.0 45.0 # Default minimum angle betw. bridge and part profile (metric imperial)
 5 5 # Default maximum allowable no of bridges per part (metric imperial)
 0.0 0.0 # Default maximum bridge length (metric imperial)

@SKELETON_CUT

750 750 20.0 20.0 # Default X-Spacing and Y-Spacing of Skeleton-Cut-Off (metric imperial)
 10 0.5 # Default value for Skeleton-bridge or offset from part profile (metric imperial)
 300 300 12.0 12.0 # Default min-X and min-Y for Skeleton Cut-Off & Shear Cut (metric imperial)
 0.5 # Nearest Cut-Off to edge of stock for clearance(range: 0-1). Factor of Spacing.
 20 0.5 # Min length of Cut-Off (metric imperial)
 0 # 0: Default; 1: To edge of Encl Rect (if partially packed)
 900 2.25 # Default min area of Scrap-pocket (metric imperial)
 9500 36.0 # Default max area of Scrap-pocket (metric imperial)
 13.0 0.5 # Default Min length of Scrap-pocket (metric imperial)
 45.0 # Maximum angle between vectors

@COMMONLINE

TOOL_PATH 3 1 5 # Default Layername, Color of Horz-Toolpath, Color of Vert-Toolpath, Color of Toekick/non-rectangle
 4.0 0.175 # Toekick - min radius for outside arc (metric imperial)
 3.0 0.125 # Toekick - min radius for inside arc (metric imperial)

@LAYOUT

1.1 0 # Horz Display Ratio between layouts (0:based on individual stock length; 1:based on longest stock length)
 40 # Report length of Layout Summary
 1.2 # Vert Display Ratio between rows (based on the biggest stock width)
 0 # Max no. of layouts displayed per row (0: no limit)
 1 # Location of Layout Summary (1:Right; 2:Top-Left; 3:Top-Right)
 0 # 1: Display part's tool profile for common-cut; 0: Do not display
 1 # 1: Show Cost in Layout & Task Summary reports; 0: Do not show
 1 # 1:Display Auto-numbering of Shear cutting sequence ; 0:Do not display
 25.0 1.0 # Min clearance (to nearest Stock)to draw Shear Cut Lines (metric imperial)
 0 # 0:Show complete list of Stocks on-screen and in reports; 1:Show Used Stocks only on-screen; 2: Show Used Stocks only on-screen and in reports

@OTHERS

1 # ViewNEST prompt - 1: Auto-Bridge Only; 2: Auto-Bridge/Common-Line
 1 0 # ViewNEST Skeleton-Cut-Off prompt - 1:Yes, 0:No; Default answer 0:None, 1:Skel-Cut-Off, 2:Scrap-pkt, 3:Both

Lines starting with # character denote 'Comments'. The 'AutoNEST V9' must be in one of the comment lines. There is no limit to the number of comment lines.

Explanatory Notes on ANEST.SET

- Line 1 and 3** These are template **DXF** files created by your particular CAD system. **DXF** support for each CAD system may differ slightly, *AutoNEST* requires these templates from sample **DXF** files to generate a fully compatible **DXF** file. Such **DXF** file should be loaded with all the necessary information, e.g. linetypes, layers, etc.
- Line 2** Refers to **DXF** format for part/shape geometry files.
(1:Block / 0: Entities)
For BLOCK representation, the part/shapes must be inserted into the drawing as AutoCAD Blocks. ALL Blocks must be inserted in the "ANEST" layer. The part/shape name will be written into the Block section of the **DXF** file for that particular part/shape. For ENTITIES format, the part /shape must be drawn using "entities" (eg. LINE, ARC, CIRCLE ...etc). Each file should contain 1 part /shape.
- Line 4** Refers to the **DXF** format of the nested output. (1:Block / 0: Entities)
DXF Option in SYSDATA
For BLOCK format, *AutoNEST* will copy the part /shape block information from the dxf file of the part/shape only once and insert the part/shape name repeatedly as required into the dxf file of the nested layout. This option will produce a smaller dxf file.
For ENTITIES format, *AutoNEST* will copy part/shape information from the dxf file of the part/shape and write these information into the dxf file of the nested layout repeatedly.
For reviewing of nested output within **Nest Manager** environment, the user can only choose either **DXF** or **VEC** File Format option in **Sysdata**.
VEC Option in SYSDATA
For BLOCK format, *AutoNEST* will read all points from the **VEC** file of the part/shape and create a part/shape block in the **DXF** file of the nested layout. This part/shape block will be inserted repeatedly as required.
For ENTITIES format, *AutoNEST* will read all points from the **VEC** file of the part/shape, then copy these points into the **DXF** file of the nested layout repeatedly when as required.
- Line 5** Denotes to the Reference Point whereby the insertion points of the part/ shapes within the nested layout are referred to.
0 - Use the centre of the enclosing rectangle of the part
1 - Use the insertion point as indicated in line 6
2 - Use the INSBASE variable value from part DXF file
- Line 6** Input of x and y coordinates.
- Line 7** 1 - assume the layer name in Line 8 when converting .VEC or .STK files to DXF format.
- Line 8** Default layer name for DXF files if .VEC or .STK files are converted to DXF format.
- Line 9** Default Layer name of the enclosing rectangle of the nested results.
If the enclosing rectangle is not required, enter an asterisk character "*" (without the quotes).
- Line 10** Default Color of the enclosing rectangle of the nested results.

Under **@TASK** are the Default values for **TaskEdit**.

Line 1 to 8 Not applicable for *AutoNEST*.

Line 9 The 1st value is the default Cutting Gap.

The 2nd value is the default Shear Blade limit.

Line 10 First 4 values - Default edge allowance for regular stock (Left, Right, Bottom and Top).
5th value - Default edge allowance for irregular stock.

Line 11 **Default values for Nest Options.** (same format as .JOB)
Nesting control parameters.
The 1st parameter is for Single Part nesting.

The 1st parameter can be:

'D' - means single part nesting with highest Density (in single array)
'M' - means single part nesting with Maximum quantity (in single array)
'C' - means single part nesting with Combination of density & maximum quantity (in mixed array)
'E' - means extension - long nest.

The 2nd parameter is to specify which type of stock to be used first if there are both Regular Stock and Irregular Stock in the same job/ task.

'R' - means nest all Regular Stocks first before any Irregular Stocks
'I' - means nest all Irregular Stocks first before any Regular Stocks

The 3rd parameter is for Packing Start Point

- 1 Left Bottom
- 2 Left Top
- 3 Right Top
- 4 Right Bottom

The 4th parameter is for Packing direction control

- 1 Horizontal Packing
- 2 Vertical Packing
- 3 Auto (System Control)
- 4 Horizontal Packing for All Stocks
- 5 Vertical Packing for All Stocks

The 5th parameter is for Common Line option

- 0 Without Common Line Consideration
- 1 With common Line Consideration

The 6th parameter is for Mirror option (Single part only)

- 0 Do NOT allow Part Mirror
- 1 Allow Part Mirror
- 2 Nest Half Quantity with Mirrored Parts

The 7th parameter is for Ignore Part Hole option

- 0 Ignore 'Hole' of part (will not nest any parts inside part holes)
- 1 Do NOT ignore 'hole' of part (will nest smaller parts inside part holes)

The 8th parameter is for Save Remnant option

- 0 Do NOT save remnant stocks
- 1 Save remnant stocks into .STK files.

The 9th and 10th parameters are for Min. Remnant Size in X and Y. Only remnant stocks that are greater than this Min. Remnant Size will be saved into .stk files onto the

default Part/ Ir-Stock directory and provided the "Save Remnant" option (8th parameter) is set to 1.

The 11th and 12th parameters are for 'Parts Priority Control'.

11th parameter is for "Max. No. of Priority per Stock" (Integer ≥ 0)

12th parameter is for "Min. Nest-able Remaining Space" percentage. (range : 0.00 to 100.00)

Line 12 Default Stock specifications for each distinct regular Stock - in the following format :-

STOCK = Width x Length Qty Priority Cost-per-Stock

"STOCK" must be in upper case or capital letters.

If the Stock is "coil", append the ">" character to the Length of the stock size.

Example : **2000 x 4000>**

If the Stock is non-coil, just give the exact dimension of the stock size.

Example : **3000 x 6000**

Qty refers to the quantity of each distinct stock available for nesting.

Priority of each distinct stock. 1 has the highest priority whereas 99 has the lowest priority. This refers to the priority for regular stocks only.

Cost-per-Stock refers to the cost for each piece of the stock.

Line 13 Default Irregular Stock specifications of each distinct irregular stock (ir-stock). Each line has the following format:

IR_STOCK = Name Qty Priority

"IR_STOCK" must be in upper case or capital letters.

Name is the .STK (ir-stock) file name. The .STK file stores the ir-stock geometry profile.

Qty refers to the quantity of each distinct ir-stocks available for nesting.

Priority of each distinct ir-stock. 1 has the highest priority whereas 99 has the lowest priority. This refers to the priority for ir-stocks only.

Line 14 **Default Multi Stock Option (Nest Option)**

- 0 Auto
- 1 Extended - Best Yield
- 2 Extended - Lowest Cost

Line 15 **Default Part Orientation**, with or without +180.

- 0 Do not support +180
- 1 Support +180

Line 16 **Default Nesting Step Angle** (Integer, range: 0 to 360). The larger the number, the longer the nesting time.

- 5 (Recommended)

Line 17 **Default Part Orientation** (0 90 180 or step 3 0 360 or ALL)

If there is more than 1 default angles, they must be separated by a SPACE.

ALL (reserved word) means all orientations are permissible.

Line 18 Default Part Pairing.

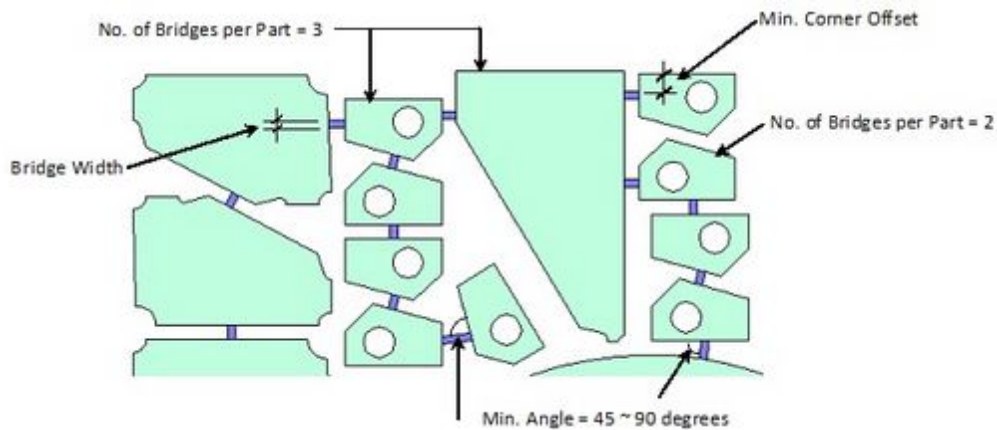
- 0 No to Pairing
- 1 Yes to Pairing (default)

Line 19 3 Default Stock Priority

- 1 Default Ir_stock Priority
- 5 Default Part Priority

Line 20 Stock Format for IMPORT STOCKS. Accepted formats: WxL or LxW**Line 21 Top-up stocks if the existing stocks are insufficient:** 1: Yes; 0: No (currently not in use)

@BRIDGE - The next 5 lines after @BRIDGE are the default bridge parameters (in Metric & Imperial English units) for Auto-Bridge. See the following illustration for explanation.

Illustration for Bridge parameters**@SKELETON_CUT**

```

750 750 20.0 20.0 # Default X-Spacing and Y-Spacing of Skeleton-Cut-Off (metric imperial)
10 0.5           # Default value for Skeleton-bridge or offset from part profile (metric imperial)
300 300 12.0 12.0 # Default min-X and min-Y (metric imperial)
0.5             # Nearest Cut-Off to edge of stock for clearance(range: 0-1). Factor of Spacing.
20 0.9          # Min length of Cut-Off (metric imperial)
1              # 0: Default; 1: To edge of Encl Rect (if partially packed)

```

These are the default values for Skeleton-Cut-Off whenever a Task is being created. Each of them are explained below :-

Line 1 Default X-Spacing and Y-Spacing.

The first 2 values are for Metric units whereas the 3rd and 4th values are for Imperial units.

Line 2 Default Skeleton-bridge or Offset from part profile

The first value is for Metric units whereas the 2nd is for Imperial units.

Line 3 Default minimum X and Y dimension.

The first 2 values are for Metric units whereas the 3rd and 4th values are for Imperial units.

When there is NO remnant specified, Skeleton-Cut-Off will use these values to ensure that re-useable space (with these X and Y dimensions or more) on the stock will be untouched by Skeleton-Cut-Off.

Line 4 Default Skeleton Cut-Off to the nearest stock edge (range 0 to 1)

This value is a factor of the Spacing (eg. $0.5 * \text{Spacing}$). Any skeleton-cut-off that is less than $(0.5 * \text{Spacing})$ from the edge of the stock will not be drawn.

Line 5 Default minimum length of Skeleton-Cut-Off

The first value is for Metric units whereas the 2nd is for Imperial units. This is to ensure that any length of Cut-off, smaller than this value will not be drawn.

Line 6 0: Default; 1: To edge of Encl Rect (if partially packed)

When the value is set to "0", skeleton-cut-off will not draw into any re-useable space that is equal or more than the Minimum X and Minimum Y (line 3).

When the value is set to "1", skeleton-cut-off will be drawn – up to the parts' enclosing rectangle.

Line 7 Default minimum area of Scrap-pocket.

The first value is for Metric units whereas the 2nd value is for Imperial units.

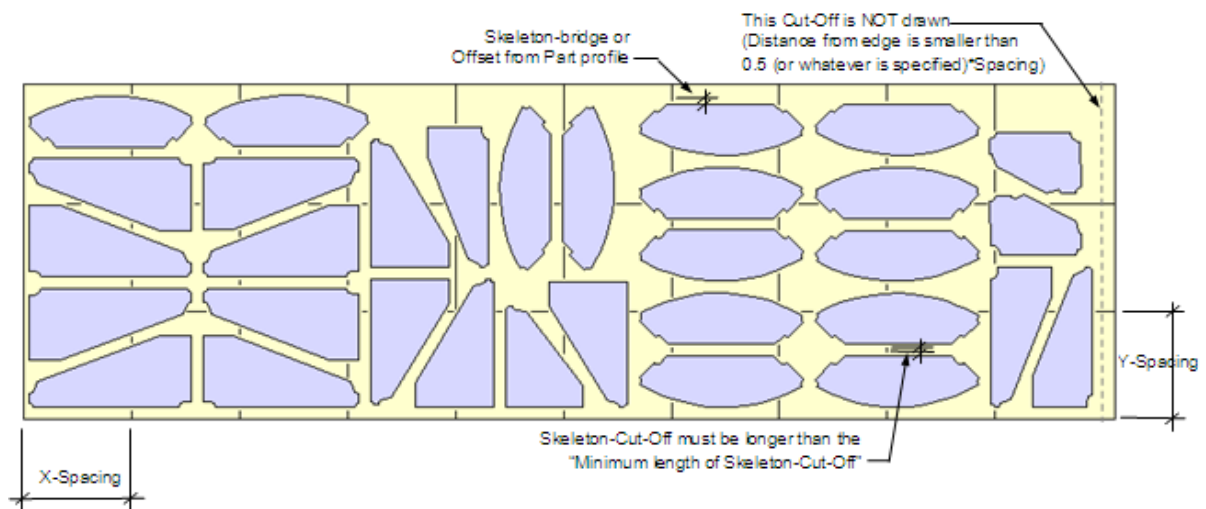
Line 8 Default maximum area of Scrap-pocket.

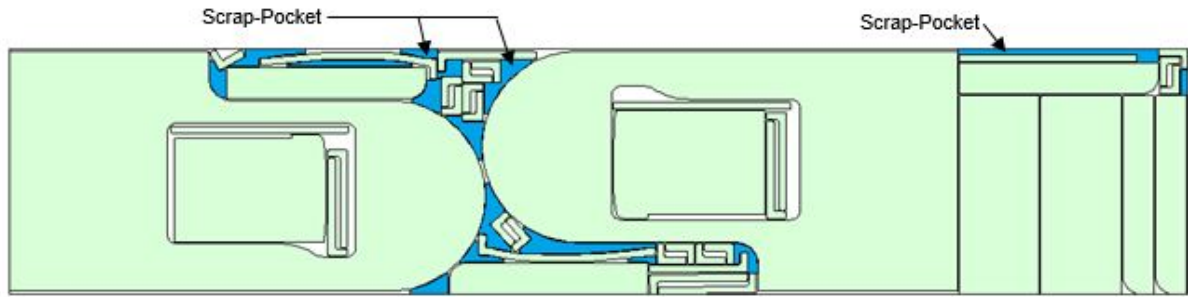
The first value is for Metric units whereas the 2nd value is for Imperial units.

Line 9 Default minimum length of Scrap-pocket.

The first value is for Metric units whereas the 2nd value is for Imperial units.

Line 10 Maximum angle between 2 vectors



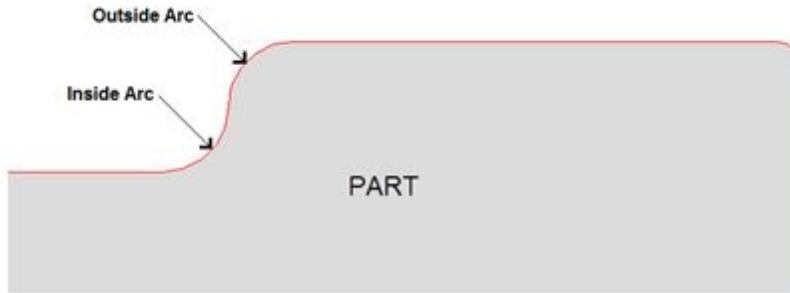
**@COMMONLINE**

TOOL_PATH 3 1 5 # Default Layername, Color of Horz-Toolpath, Color of Vert-Toolpath, Color of Toekick

/non-rectangle

4.0 0.175 # Toekick - min radius for outside arc (metric imperial)

3.0 0.125 # Toekick - min radius for inside arc (metric imperial)



@LAYOUT - The parameters under @LAYOUT affect how nested results are displayed.

1.1 0 # Horz Display Ratio between layouts (0: based on individual stock length; 1: based on longest stock length)

40 # Report length of Layout Summary

1.2 # Vert Display Ratio between rows (based on the biggest stock width)

3 # Max no. of layouts displayed per row (0: no limit)

1 # Location of Layout Summary (1:Right; 2:Top-Left; 3:Top-Right)

0 # 1:Display part's tool profile for common-cut; 0 :Do not display

1 # 1:Show Cost in Layout & Task Summary reports; 0 :Do not show

1 # 1:Display Auto-numbering of Shear cutting sequence ; 0:Do not display

13.0 0.5 # Min clearance (to nearest Stock)to draw Shear Cut Lines (metric imperial)

0 # 0:Show complete list of Stocks in Task Report; 1:Show Used Stocks only;
2: Show Used Stocks in Task Report & .sum

Line 1 to 5 Parameters that control the display of nested layouts. See the illustrations A, B and C below for an explanation of these parameters.

Line 6 This parameter controls the display of parts (tool-profile) in the nested layout

0 Display Part as per normal

1 Display Part expanded with half cutting gap.

Line 7 This parameter determines the display of COST of stocks on the nested layout

0 Do NOT show the Cost.

- 1 Show the COST of stocks.

Line 8 This parameter controls the display of the **Auto-Numbering of Shear** cutting sequence on the nested layouts. (see Illustration below)

- 0 Do NOT display
- 1 Display auto-numbering

Line 9 Applicable for Shear Cutting Lines. This is the **minimum clearance** from the edge of stock to the cutting line in order to draw the cutting line. If the distance is deemed to close to the stock (less than min. clearance), then no cutting line will be drawn.

The first value is for metric units, the second is for imperial units.
See Illustration below.

Illustration A (based on the above first 5 parameters @LAYOUT)

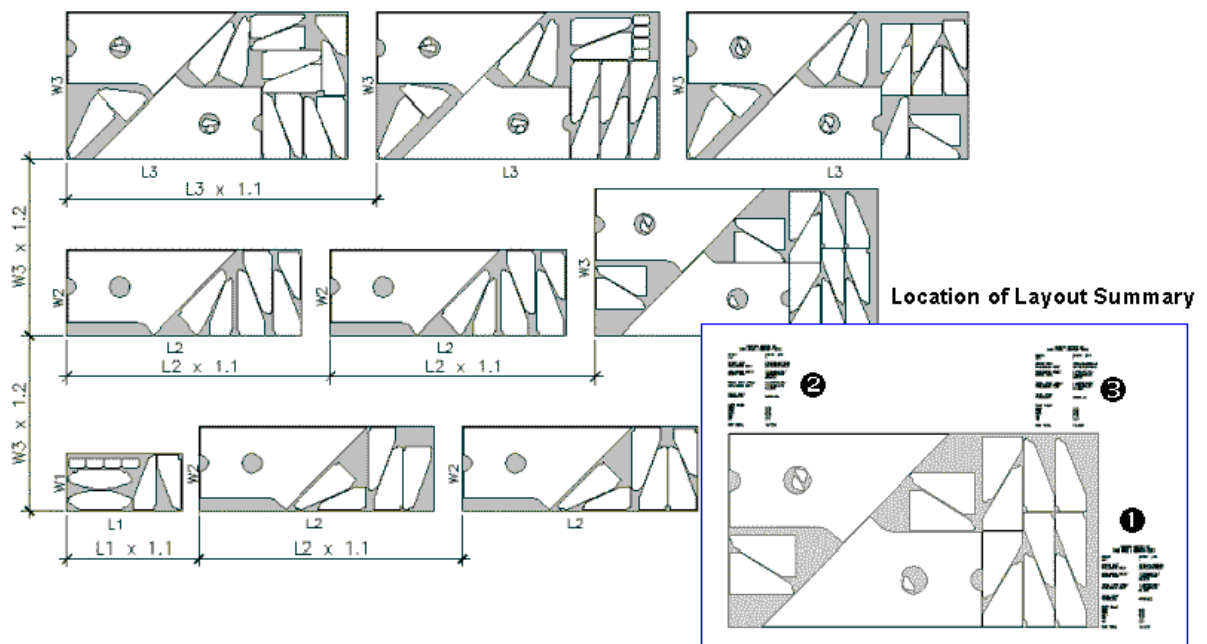


Illustration B - Layout Summary report at "Location 1"

Textsize : Layout Summary textsize as defined in **Sysdata**. (8th line in ANEST.ARG)

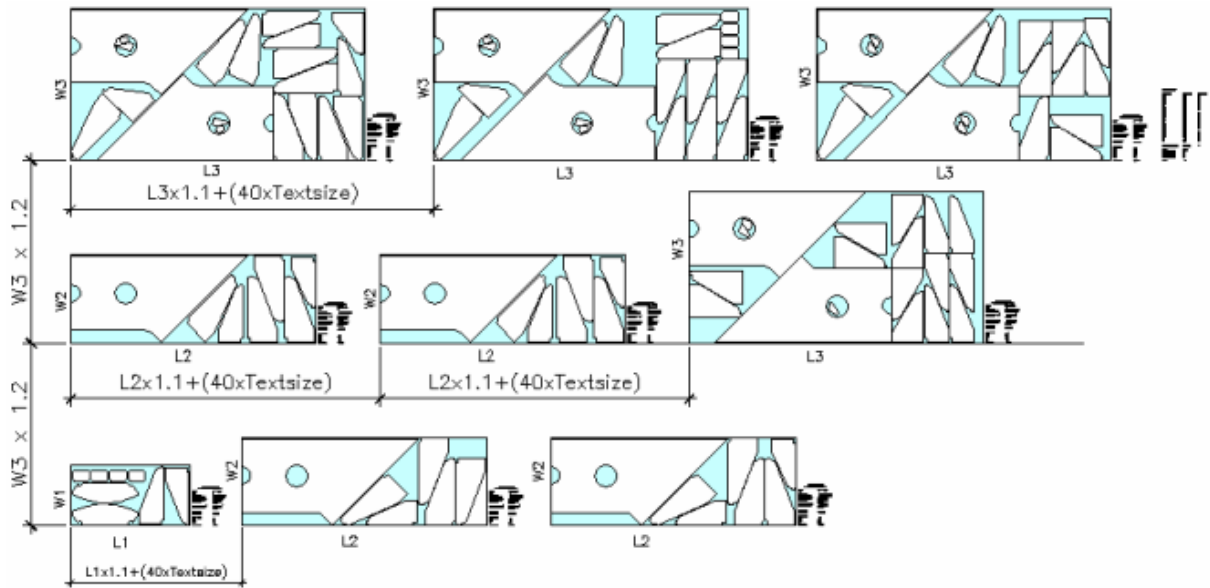
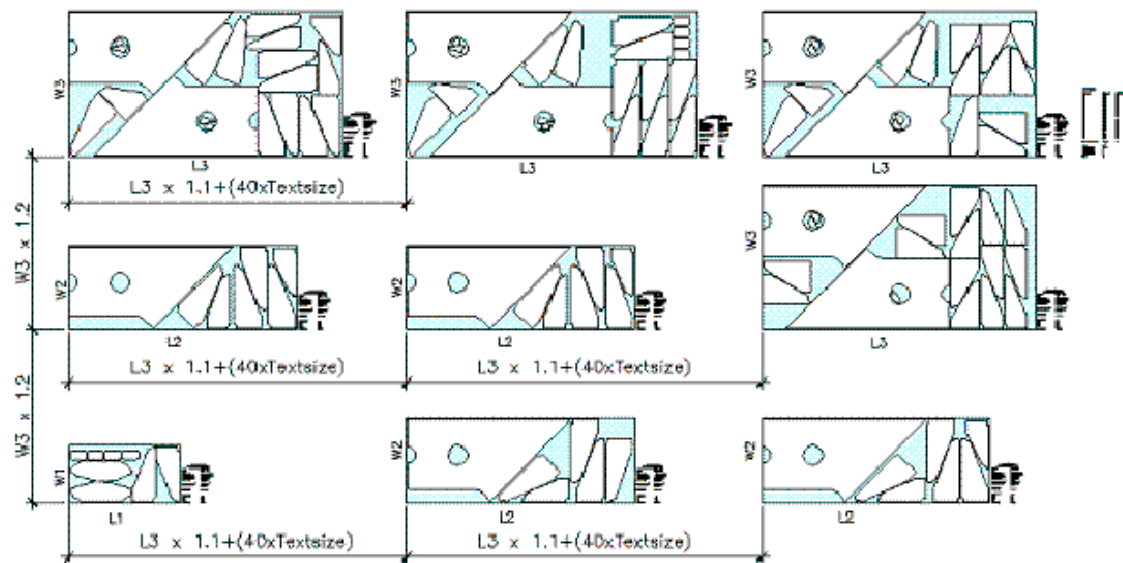
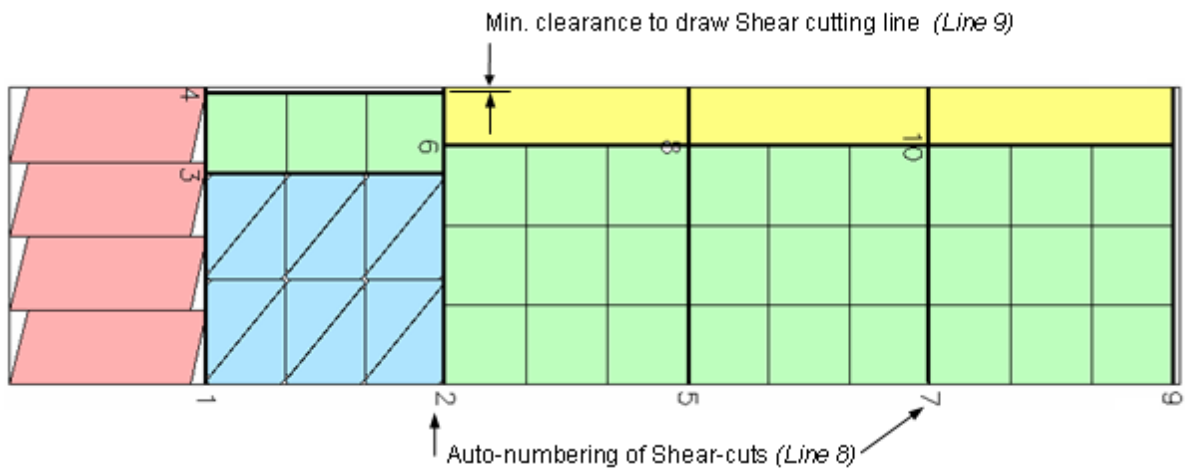


Illustration C - Horz display between layouts based on longest Stock Length
@LAYOUT

1.1 1 # Horz Display Ratio between layouts (0: based on individual stock length; 1: based on longest stock length)



Shear Auto-Numbering & Minimum Clearance (Illustration for Line 8 & 9)



Line 10 This is to control the on-screen display of the list of Stocks in the reports.

- 0 Show complete list of Stocks on-screen and in .SUM report.
- 1 Show Used Stocks only on-screen.
- 2 Show Used Stocks only on-screen and in .SUM report.

@OTHERS - This is the prompt options for the ViewNEST command in the AutoCAD environment.

The 1st line controls the prompt for Auto-bridge and Commonline(Toolpath).

The 2nd line controls the prompt for Skeleton-Cut-off.

Advanced Programmer Topics

Advanced Programmer Topics deals with the more advanced features that are in Router-CIM. These topics should be used by users that are familiar with Router-CIM and the topics that are discussed.

- 1) [Post Processor](#)
- 2) [Video Jet Print Head](#)
- 3) [Router-CIM Acceleration/Deceleration](#)
- 4) [How to Backplot NC Code](#)

Advanced Programming - Post Processor

A "post processor" is a computer program which is used to translate data from one format to another format. The postprocessor adapts computer-aided design data format, output by Router-CIM and AutoCAD programs, to a particular NC machine tool format. The NC machine format is actually a format required by a specific machine-tool/control-unit combination. This is necessary because each machine tool has its own characteristics.

The rules and regulations that govern the translation from one data format to another is provided by an international standard named EIA-RS-274-D. The standard represents a common method of translation of each machine activity into a special code for the machine tool. The standard allows for optional codes to be defined so that machine tool manufacturers have latitude in choosing machine codes.

The postprocessor translates AutoCAD data (.DXF) into M and G codes (EIA-RS-274-D) using the basic principals provided in the international standard. Additional translation methods are provided to address translation requirements that vary from the standard.

NOTE: You should only change post processor files after consulting CIM-Tech. An improper setting/change could cause a failure in Router-CIM that may be hard to track down or troubleshoot.

Refer to the following sections for more information:

1. [Post Processor Considerations](#)
2. [Post Processor Output](#)
3. [Using the Router-CIM Post Processor](#)
4. [Cut and Sequence](#)
5. [Processing Concepts](#)
6. [Connecting the Dots](#)
7. [Configuring the "\\$PP" File](#)
8. [Translation of Text Statements from a Drawing](#)
9. [Format of AutoCAD Text Statements](#)
10. [Translation of Geometry from a Drawing](#)
11. [Motion Block Definition](#)
12. [Group Definition and 'G' Switch](#)
13. [Absolute Preset Example](#)
14. [Pocketing Example](#)
15. [Switches](#)
16. [Numeric Formatting](#)
17. [Global Numeric Formatting](#)
18. [Individual Numeric Formatting](#)
19. [Modals](#)
20. [ASCII Modals](#)
21. [NUMERIC Modals](#)
22. [AXIS Modals](#)
23. [Reserved Words](#)
24. [Fine Tuning](#)
25. [Switch Parameters](#)
26. [General Format of ;P Switch](#)
27. ["F" Switch Parameters](#)
28. [Geometric Entries](#)
29. [Customization](#)

Post Processor Considerations

The process of translation is a sequential process. Each time a piece of data (such as a line or arc element) is encountered, a specific NC code is developed for that piece of data. This means that the data to be translated must be in the order in which the elements are to be processed. Router-CIM provides the necessary commands to ensure that proper order of the data is achieved.

One printed line of NC code is referred to as a command block or motion block. An example would be the line "N1G01X3Y3M08". Within this command block, several NC commands exist. In this case the M08 turns on an air blower unit and the G01X3Y3 performs a move to the location 3,3.

Several NC commands can exist in one command block. Methods are provided to define the different NC codes that should appear in one command block. The block definitions discussed in the "Configuration" section refer to this grouping of NC codes into one command block.

The translation rules and the arrangement of codes within a command block is governed by an ASCII file. This file has a file extension of .\$PP. This file represents the heart of the postprocessor. Each line of information in this file contains the translation rules and the NC code grouping requirements. Any behavior changes in the translation process can usually be affected by changing the \$PP file.

Various other files affect the translation of codes to the proper format. Some of the other files in addition to the \$pp file are:

Postname.tsl

Postname.tsk

Postname.seq

Postname.dat

Postname.wiz

Postname.lsp

Postname.fas

Postname.dwg

Postname.bi

Postname.dcl

These files exist in the Ncpost and Ncdwgs folders within Router-CIM's directory structure and perform functions that work together with the \$PP file to achieve the proper translation.

Other files may exist in particular systems, depending on the complexity of the machine and the amount of modification required to the Router-CIM system to achieve the proper translation.

Router-CIM works in a fairly straight forward manner, and according to some strict rules that govern all the steps that are made in the software from the time you start it, until the time you make code. Understanding these steps is critical if you wish to modify the output to suit a particular machine tool.

Post Processor Output

The output product is a file that represents a part program. Actually, two files are produced. The files have different extensions, usually .PSC and .OUT. The file known as the .PSC file is a ASCII representation of the part program.

This file can be edited by any text editor. The file known as the .OUT file is either in ASCII or EIA format and can be shipped (via a user supplied communications package) to a machine tool controller through an RS232 cable or in some cases, a direct network connection.

The .PSC output file is the results of translating an AutoCAD .DXF file into NC code. The .DXF file is normally developed with the help of Router-CIM to ensure proper order of the drawing data.

The .OUT output file is the results of converting the .PSC file into NC codes that will be transmitted to the machine tool. Special handling occurs to ensure proper transmission codes, end of block codes, and character (EIA or ASCII) codes. This is the main output file you are concerned with as it is the file ultimately sent to the machine tool for processing.

Using the Router-CIM Post Processor

Post processing occurs after all the elements and text information in the AutoCAD drawing have been described and Router-CIM has ordered and sequenced these elements. Post processing is the last thing done in the development of NC code.

Post processing is the byproduct of the Sequence command that converts an AutoCAD .DXF file into NC code. The required .\$PP file is provided so that the translation of AutoCAD data to NC code is done without regard to necessary rules and format.

The postprocessor communicates with AutoCAD via the AutoCAD .DXF file. This file represents the current drawing data in an AutoCAD session at the time an AutoCAD DXFOUT command is given. In order for the postprocessor to translate AutoCAD drawing data, a .DXF file must exist.

As a user, these commands are transparent to you, and happen fairly quickly once the Sequence command is given.

If the postprocessor is properly set up, there should be no need for extra editing of the NC Code file once it is created.

There are several methods in Router-CIM for inserting commands into the NC Code file before it is created, and those will be discussed briefly at first and then in more detail later in this reference.

Cut and Sequence

There are two distinct steps in Router-CIM. The cut step and the Sequence step. As outlined above, post processing is a pretty linear step. It takes all the elements in order during Sequence and outputs them according to a set of rules defined in the system, passes them through the \$pp file and then into an NC Code file.

But what gets all the bits into the drawing so that they can be output during the post processing step? The answer is Cut, and Sequence.

After your geometry is geoshaped, and all the necessary knowledge information has been filled in, such as tool number, feed, speed, depth of cut, then you press cut, and a toolpath appears on your screen in accordance with your knowledge parameters. These cuts have an abundance of data stored in them, and contain many of the pieces that the post processor will later need in order to properly generate the NC Code.

So, how does all the data get into the cut? The answer is tasks. Tasks are small programs, written in a form of Autolisp. Kind of like a macro. These tasks place text words, and sometimes geometry into a toolpath. It is necessary sometimes to make changes to tasks, so that the right words are embedded into your cut, long before you get anywhere near the post processing stage.

Each post processor is usually shipped with a set of tasks that can be modified to allow changes to the behavior of the cut, and therefore to the behavior of the NC Code that is developed later. These are the .tsk files that are named with the same name as the post in the Ncpost folder.

As important as it is to have specific words in a particular toolpath, sometimes you need words that are generic to the toolpaths, but specific to a particular machine. What I mean by that is a word that needs

to be output in a program, but not necessarily in any one cut. Maybe at the beginning of the program or the end, or even in between one cut and another.

To accomplish this feat, we use the Sequencer. The Sequencer is a set of files that get run when you press the Sequence button in Router-CIM. When you do so, you are prompted to select the toolpaths that you want turned into NC Code. The Sequencer then looks up all the things it has to do, and in what order. The Sequencer then performs its functions, one of which is to run the program that steps all the words through the \$pp file and turns them into NC Code.

The Sequencer can, similar to tasks, be modified on a per machine basis to change the way the code is output. It too uses Autolisp, and has several programs in its files, usually called Defuns (define function) to do its work. If you needed words or behavior to happen at the beginning of every program, you use the Defuns that run at the beginning. Similarly there are Defuns to govern what happens during a toolchange, or even what happens when you switch from one cut to another cut with the same tool.

The Sequencer files are the .seq and the .tsl. These files are in the Ncpost folder along with the rest of the files for a particular machine.

There will be more on the tasks and Sequencer files later in this book.

Processing Concepts

The postprocessor translates certain AutoCAD drawing elements. When an AutoCAD .DXF file is created by AutoCAD, every type of element currently in the drawing is in the .DXF file. The Router-CIM postprocessor will filter out the AutoCAD elements that do not have meaning to the NC translation process. Due to the large list of element types available in AutoCAD, the following list contains only those data elements that the postprocessor will process. AutoCAD drawing elements processed by postprocessor are:

Line

Circle (later broken up into an arc)

Arc

Point

Polyline (later broken up into arcs and lines)

Spline (later broken up into lines or arcs)

Ellipse (later broken up into lines or arcs)

Text

All other AutoCAD drawing elements are filtered out during the post processing step and ignored.

During the Geoshape step in Router-CIM the geometry elements listed above are reduced to their smallest elements. That means that a polyline, for instance, will be broken down into lines and arcs as

necessary for the translation into NC Code. Since most machine tools will only be capable of making either a linear or arc motion, all elements eventually turn into these geometric pieces.

The postprocessor translates AutoCAD .DXF data in a specific manner. This manner of processing AutoCAD data to NC code is outlined in this section.

The process of translation is a sequential process. Each time a piece of data is encountered, a specific NC code is developed for that piece of data. This basic concept is applied to every AutoCAD element in the input .DXF file.

Since NC code is based on motion blocks (sometimes referred to as command blocks), the "trigger" that outputs NC code is a motion by the machine tool. This motion is any X, Y, or Z AutoCAD entity, such as a line, arc, circle, polyline, spline, ellipse or point. Even though the motion may not physically move the machine tool, a command block is developed. An AutoCAD data element that is considered a tool motion but does not physically move the machine is a point at a location where the machine tool is currently located.

A text line of NC code (command block) is produced every time the postprocessor encounters a physical AutoCAD element. A physical element is a line, point, arc, circle, polyline, ellipse, or spline.

AutoCAD text statements are "collected" by the postprocessor and then output as NC code when a physical AutoCAD element is processed.

Understanding the two facets of processing outlined above, we can combine text statements with AutoCAD geometry to yield certain results. An example would be two text statements followed by a line element.

These three AutoCAD elements would be in the order of TEXT, TEXT, and LINE. These three elements would produce one text line or motion block of NC code. The postprocessor would sequentially "collect" the two text statements, then encounter the line element. The line element would cause an NC text output. The format of the command block with the two text statements and the line is governed by the .\$PP file.

This section only touches on the concept. The "Configuration" section provides a technical description of the various formatting features of the program.

Connecting the Dots

The postprocessor supplies additional AutoCAD elements into the input .DXF file. Since it is a sequential processor, all geometry elements MUST be connected to each other. This is done because a machine tool travels from the end of one element (current position) to the start of the next position (goto position). "Connecting the Dots" means that everywhere a geometric "GAP" in the sequential AutoCAD data occurs a line element is developed. This line element bridges the GAP between the end of the last element to the start of the next element. This behavior is commonly known as INDEXING from one element to the next.

This additional line element will affect the processing of the AutoCAD .DXF file. When defining text statements at locations that are at the beginning or end of geometries that are not physically connected, remember the following:

The postprocessor develops a line between all geometries that have a physical gap. This developed line is the same type of line element as though it was developed in AutoCAD. Any text that appears at the beginning or end of this developed line will appear in the NC code according to the rules outlined above. That is, a text statement will be collected and then output when a physical element (such as a line) is encountered.

Configuring the "\$PP" File

The "Configuration" section provides a technical description of what things can be changed in the . \$PP file to effect the translation behavior of the postprocessor. The method of changing this file is through the use of a text editor. The text editor must not produce additional control characters. Most word processors require special characters to control formatting. These word processors may also have a mode to suppress these special characters. This mode is known as "programmer's mode". Any text editor that **does not produce special characters** can be used to edit the . \$PP file. The most common is the program named NotePad. It exists in all versions of Microsoft Windows and does not produce special characters. Be sure, however, to turn off Word Wrap as it may format the code improperly.

If more powerful tools are needed, We recommend a program called UltraEdit as it is capable of using macros and has syntax highlighting that will allow you to easier format a file for proper editing.

The postprocessor is a program that reads an AutoCAD drawing exchange file (.DXF) as input and produces numerical control code as output. This translation of a drawing into NC code is controlled by an ASCII text file referred to as the \$PP file (the dollar p-p file). This file is usually named after the particular machine tool controller that is being used. Example: Mach1s.\$PP or GE550T.\$PP

The \$PP file is tailored for each particular machine tool/controller combination. It is usually shipped to the user already configured. If you received a preconfigured system, then you may wish to skip this section. It is only provided for the individual who wants to configure the \$PP file.

Each line or entry in the \$PP file defines a specific translation to the post. At the simplest level an entry consists of NC code on the left of an "=" sign, and the AutoCAD text on the right side of the "=" sign.

Example:

M30 ; A=FINI

This entry in the \$PP file indicates that when the text "FINI" statement is processed by the post, it will be translated to "M30" when it is output to the ".NC" file.

Each entry to the right of the "=" sign must be unique. No two text strings on the right of the "=" can be the same.

In the following examples, characters that are boldfaced represent separator or delimiter characters that are used to separate postprocessor information. Throughout the \$PP file, the characters '#' and ';' are used to separate switches and strings to control post operation.

Each line or entry in the "\$PP" file is structured in one of the three following formats:

1. NC Code Format ; Switches = Text from drawing.

2. NC Code # Format ; Switches = Geometry from drawing.

3. \$PP entry # \$PP entry = \$ Blocktype.

The next three sections explain these different types of postprocessor entries and how they are used to formulate NC code.

Translation of Text Statements from a Drawing

The postprocessor can translate APT-like text statements from the drawing into NC code. The general format of the entry in the \$PP file that is used to translate text is as follows:

NC CODE # FORMAT ; SWITCHES = TEXT

In the above format "#", ";", and "=" are used as separators.

The NC CODE portion of the \$PP entry can be a single character or a string of characters that is output as NC Code. Typical codes might be as follows:

G90

G92 X0000.000 Y0000.000 Z0000.000

The FORMAT portion of the \$PP entry allows for formatting of the numeric value in the text statement. The format specifies the number of places to the left of the decimal point and the number of places to the right of the decimal point. The first format is used when the postprocessor is in ENGLISH mode (the default). If a second format is encountered, it indicates the format to be used when the postprocessor is in METRIC mode. Each format is separated by the delimiter #. (Most machine tools use different formats for English/Metric code). A typical format would be as follows with 3.4 for ENGLISH and 4.3 for METRIC:

#3.4#4.3

The SWITCHES portion of the \$PP entry allows for a wide variety of options concerning formatting and positional placement of the NC Code within the current motion block.

The TEXT portion of the \$PP entry that is on the right side of the equal(=) sign is the key that is used to look up and translate the text to NC code. Every text statement in the AutoCAD drawing must have an entry in the \$PP file. Each text statement is searched for in the \$PP file. When a match is found, the postprocessor translates the text statement into NC code using the parameters described on the left side of the equal(=) sign.

Format of AutoCAD Text Statements

The text statement in an AutoCAD drawing is structured as follows:

TEXT / Number , Parameters

Examples of valid NC text statements are:

ABS

FEDRAT/10

M40,U10,V12

FEDRAT/10,M42,V12

Text can be any CAPITALIZED TEXT defined on the right hand side of the "=" sign in the \$PP file. Number is an optional numeric value. Parameters are simply text statements separated by commas that are passed through to NC Code unaltered by the postprocessor. Since Parameters are passed unaltered, they MUST BE entered with the appropriate format. For example, if X1 and Y1 are passed as parameters, they must appear as X1000Y1000 for the controller to read them properly, if the controller requires trailing zeros.

Translation of Geometry from a Drawing

The general format of a \$PP entry that controls the translation of geometric information from the drawing into NC code is as follows:

NC CODE # FORMAT ; SWITCHES = GEOMETRY

This format is identical in use to the text control lines in the previous section except that the codes will be output based on geometry instead of text statements.

The GEOMETRY portion of the \$PP entry indicates that a specific value from the geometry in the drawing is to be formatted and output as NC code. The geometric entries that are predefined to the post include standard X,Y, and Z values as well as other computed values. These entries are set based on the piece of geometry that is being processed (line, point, arc).

For example, if the \$PP file contains the entries "X#4.3=X" and "Y#4.3=Y", then any X or Y coordinates from the drawing will be output with a format of "4.3". When a line from 0,0 to 2.2222,3.33333 is processed by the post, the NC code "X2.222 Y3.333" will result. If "3.4" had been the format, then "X2.2222 Y3.3333" would result.

As another example, if the \$PP file contains the entries "Z#4.3;=-X" and "X#4.3=Y" and a line from 0,0 to 2.2,3.3 is processed by the post, the NC code "Z-2.2 X3.3" will be output.

Note that this example results in a transposition of axes, as well as a sign reversal of X coordinates that is necessary in some applications.

Motion Block Definition

A postprocessor \$PP entry controls the output of the motion block information. A "motion block" is just a single line of NC code.

The following example is considered a motion block:

N1G01X1Y1\$

The motion block definition is processed and a line of NC code is produced when a specific drawing entity is processed. The general format of a motion block definition control line is:

\$PP ENTRY # \$PP ENTRY = \$BLOCKTYPE

The \$PP ENTRY portion of the definition can be any other valid entry in the \$PP file except another motion block definition. As many \$PP entries as will fit on one line can be placed in a motion block definition. Each item must be separated by "#", for formulation of the motion block.

There are two predefined \$PP entries, "PA" and "PB", which show the placement of the text statements with "A" switches and "B" switches.

The \$BLOCKTYPE portion of the motion block definition is one of the following:

\$LINE1.1

\$POINT1.1

\$TEXT1.1

\$ARC1.1

Example:

SEQNO#PB#LINEAR#RAPID#X#Y#Z#PA=\$LINE1.1

Each of the \$BLOCKTYPE strings corresponds to a geometric entity that is read from the .DXF file and processed by the postprocessor. The above example tells the post that when a line is read from the input .DXF file some geometric entries such as length, start point, etc. are computed from the lines geometry. Then the "\$LINE1.1" motion block is output to the NC file. This output consists of each of the \$PP entries defined on the left of the '=' (except for codes which have been disabled the 'D' switch).

Group Definition and 'G' Switch

The numbers on the end of the blocktype have special significance. Each blocktype ends in a "n.m" number where 'n' or the first number, called the "group" number, indicates a particular group for this definition. This can be used in conjunction with the 'G' switch to allow totally different blocks of NC code to be generated from the same geometry. The 'm' or second number indicates a sequence number for output so that more than one line of NC code may be output for a single geometric entity. If a controller needs a two line arc definition, "stuff ... = \$ARC1.1" and "stuff ...=\$ARC1.2" definitions would be needed.

Typically these definitions will be configured beforehand and will not require modification.

For example, the \$PP file contains the following entry:

SEQNO#PB#LINEAR#RAPID#X#Y#Z#PA=\$LINE1.1

If a line is processed, the \$PP entry "SEQNO" will be output to the NC file, followed by text with "B" switches, followed by the \$PP entry "LINEAR" (if enabled), followed by "RAPID" (if enabled), followed by output for X, Y, Z, and text with "A" switches. (Reference TABLE 3.1. See switches 'A', 'B', and 'C'. See switches "D" (disable) and "E" (enable) for a discussion of how "LINEAR" and "RAPID" are implemented.

Absolute Preset Example

As a detailed example of group code usage, in the ROUTER.\$PP you will notice a line which reads "G92;B;G#2#1=PRESET".

The ;G switch on this code will set the current group number to 2 when the text statement "PRESET" is processed.

This second group definition is used for PRESET and several other codes because of the modal nature of X, Y, and Z values. Modal values only output when the value changes. ROUTER.\$PP is configured so that X,Y, and Z are modal and will only output when necessary.

This creates a problem when issuing a G92, because the machine tool controller must have both X and Y to honor a G92 request.

Example of incorrect output:

Line from 0,0 to 2,2 : "N7 G01 X2 Y2" output to the NC file

Text "PRESET" : "G92" stored in PB buffer

Point at 2,3 : "N8 G92 Y3" output to the NC file (**ERROR!**)

The default postprocessor (ROUTER.\$PP) is set up to handle this problem with a different group definition for G92 output. The G92 code will switch the group number to 2 for non-modal output of X, Y, and Z.

Below the PRESET entry in the \$PP file, there is a line that reads:

```
SEQNO#PB#XNON#YNON#ZNON#PA=$POINT2.1
```

This line defines group 2 output for points. The only difference in group 2 point output and group 1 (\$POINT1.1) output is that group 2 is non-modal, and will always output values to the NC file.

With ROUTER.\$PP the above example will generate the correct NC code:

Line from 0,0 to 2,2 : "N7 G01 X2 Y2" output to the NC file

Text "PRESET" : group switched to 2, "G92" stored in PB buffer

Point at 2,3 : "N8 G92 X2 Y3" output to the NC file, group switched to 1

Pocketing Example

For another example, assume that you want to generate a pocket milling cycle from the geometric information on a line. Since lines are set up to generate a G01 or LINEAR move, a second motion block definition is needed.

The \$PP file would need the following entries:

```
SEQNO#PB#LINEAR#RAPID#X#Y#Z#PA=$LINE1.1
```

```
SEQNO#POCKET#XI#YI#PDEPTH=$LINE2.1
```

```
G00;D#LINEAR;E#RAPID=RAPID
```

```
G01;D#RAPID;E#LINEAR=LINEAR
```

```
X#3.4=X
```

```
Y#3.4=XI
```

```
Y#3.4=YI
```

```
G72;G#2#1=POCKET
```

```
Z#3.4=PDEPTH
```

The 'G' switch on the G72 code causes the postprocessor to switch to group number 2 for NC code output. Additionally the code also causes the group number to change back to 1 when the code is output to the NC file. This provides the ability to output a pocket cycle from a line and then switch back to the normal G00, G01 interpretation of a .DXF line. If you want to output a pocket cycle for each line, define the switch as "G#2#2", which causes the postprocessor to use group 2 definitions until switched back. You can build a code to switch back to group 1 by adding the entry:

```
;G#1#1=GROUP1
```

to the \$PP file. Whenever the text statement "GROUP1" is processed, no output will be generated, but the 'G' switch will still be implemented changing the group number to 1.

Example:

DXF file contains	NC code produced	Comments
Text: "LINEAR"	None	"RAPID" disabled
Line: 0,0 to 1,1	N001G01X1.Y1.	Output NC code
Line: 1,1 to 2,2	N002G01X2.Y2.	Output NC code
Text: "PDEPTH/0.05"	None	0.05 saved

Text: "POCKET"	None	"G72" saved, group switched to 2
Line: 2,2 to 3,3	N003G72X1.Y1.Z0.05	Output NC code
Line: 3,3 to 4,4	N004G01X4.Y4.	Output NC code

Note that even though G01 line output is in absolute mode, the group 2 definition for a line uses the incremental geometric entries "X1" and "Y1" in order to form the correct values for the X,Y pocket milling cycle.

Switches

Use the optional switch parameter when additional definition is required for the usage of a controller code. The switches can control positional placement, numeric formatting information, and other options specific to a particular code.

The switches follow the ";" in the \$PP entry and have the general format:

```
;SWITCH#PARAMETER#PARAMETER;SWITCH;SWITCH#PARAMETER
```

SWITCH can be any of the switches listed (separated by ";") and PARAMETER is an optional parameter that is used by the switch (separated by "#").

Example:

```
C;D#LINEAR;E#RAPID#MYSTUF
```

When a text statement from the drawing is processed, switches can determine placement of the NC code. When a "A" or "B" is encountered, the NC code is set aside in internal storage areas known as buffers. The NC code is placed sequentially in these buffers and held there until a motion block definition is processed. Note that a switch must be specified since there is no default for text entries in the \$PP file.

In the case of the "B" switch, the command is placed in the "before" buffer or the "PB" entry. In the case of the "A" switch, the command is placed in the "after" buffer or the "PA" \$PP entry.

If a "C" switch is encountered, the NC code associated with this entry is placed in the "PC" entry the motion block definition for "\$TEXT1.1" will be output to the NC file as a line of NC code. This block definition for text is normally as follows:

```
SEQNO#PB#PC#PA=$TEXT1.1
```

For example, assume the \$PP file contains the three entries:

```
G90;B=ABS
```

```
F#3.2;A=FEDRAT
```

```
G04;C=DWELL
```

If a drawing with the text statements:

```
ABS
```

```
FEDRAT/19.234
```

```
DWELL
```

is processed, the postprocessor outputs:

```
N1 G90 G04 F19.23$
```

to the NC file.

Numeric Formatting

The appearance of numbers may be changed in a wide variety of ways by the post. This control over numeric format is needed to service a wide variety of controllers.

The format of numeric output is controlled by two different methods. The first method of controlling numeric output is in the header codes of the \$PP file. The second method is a case-by-case basis for each number entry.

Any format string can be used to control the output of all numbers in the postprocessor.

Global Numeric Formatting

When numeric format is controlled by a header code, it affects all numbers output by the post. If you want FIXED OUTPUT and NO DECIMAL POINT in ALL numeric output, the following lines would be needed in the \$PP file:

```
FIXED/LEFT=$HEADER.1
```

```
FIXED/RIGHT=$HEADER.2
```

```
DECIMAL/OFF=$HEADER.3
```

Individual Numeric Formatting

The other method of format control is applied on a code-to-code basis. This format control is not applied to any other code in the \$PP file. Any format string that can be specified in the header can also be specified in the 'F' switch applied to the code.

The same format control in the above example could have been applied to a single code as follows:

```
Z#3.4;F#LRD=X
```

This defines a letter code of Z to be output followed by a numeric value obtained from the current X value.

This numeric value will be formatted with 3 places to the left and 4 places to the right of the decimal point. Further formatting control has been indicated by the presence of the 'F' switch with its parameter string LRD. Each character in the parameter list specified:

FIXED/LEFT (L), FIXED/RIGHT (R), DECIMAL/OFF (D).

If the number 2.34 was in the .DXF file the following formatted number would be produced using the . \$PP file entries outlined above:

234

The presence of the 'F' switch will cancel any formatting controls specified by \$HEADER codes.

Modals

Modals are defined as codes that only need to be output once to put a machine tool controller into a particular mode. An example would be a typical cutting mode. Once the controller is given a G01 code for cutting, it only needs X and Y codes for subsequent linear motions to be generated. If the machine mode needs to be changed then another modal code is supplied.

Modals can also be referred to when addressing machine movements. An example would be an X and Y motion. Once an X or Y location is established by the controller, the X or Y value does not have to be addressed again until a change in the value occurs. If the next motion of the machine has a different Y location and the same X location, only the Y location needs to be specified.

When a modal is referencing a machine mode, the modal is referred to as an ASCII modal because it is comparing letters. When a modal is referencing a machine motion, the modal is referred to as a NUMERIC modal because it is comparing numbers.

A simple example of \$PP file entries would be:

```
G01;M#0=LINEAR
```

```
X#3.4;M#1=X
```

```
Y#3.4;M#1=Y
```

M#0 represents ASCII modals and M#1 represents NUMERIC modals.

A simple example of how these \$PP entries effect NC code is as follows:

DXF file contains	NC code produced	Comments
Text: "LINEAR"	none	G01 mode set
Line: 0,0 to 1,1	N001G01X1.Y1.	Output mode, X and Y motion
Line: 1,1 to 2,1	N002X2.	Output X motion (Y the same)

ASCII Modals

This type of modal is referred to as modal type 0 in the postprocessor. This type of modal is appropriate for most codes represented in text. The general format of an ASCII modal code is:

```
NC CODE;M#0#ENTRY#ENTRY... = TEXT
```

Some modal G-codes can cancel other modal G-codes. The most common example is the G01 modal code. After a circular arc code G02 or G03, the G01 must be reissued to the controller. In effect, the G02 or G03 cancel the G01 modal code in the controller.

This disabling of a modal code is controlled by the 'M' switch on the G02 and G03 codes:

```
G02;M#0#LINEAR#RAPID=CIRCUL/CCLW
```

tells the postprocessor that when a counterclockwise arc is processed, G02 is modal and cancels out the modal definitions for the \$PP entries "LINEAR" and "RAPID". The \$PP file ROUTER.\$PP uses this standard definition.

Example:

NC Code	Comments
G01 X3. Y4.	Cut a line to 3,4
X2. Y2.	Cut a line to 2,2. G01 not needed
G03 X5. Y5. I1. J1.	Cut an arc. G01,G00 canceled

G01 X9. Y9.

Cut a line to 9,9. G01 needed

NUMERIC Modals

This type of modal is referred to as modal type 1 in the postprocessor. This type of modal is appropriate for most numeric input. The general format of a NUMERIC modal code is:

`NC CODE;M#1=TEXT`

Since NUMERIC modals deal with numbers, the meaning of the number can effect the results of the NUMERIC modal. A number can represent an incremental or absolute motion. Most machine tools consider a motion of 0 in incremental mode to be a modal condition. In incremental mode the value is only output when it is non-zero. This type of value oriented modal will function differently in incremental mode.

The distinction between the two types of modals is important. If the line "F#3.4;M#1=FEDRAT" is used in the \$PP file, then feedrates of 0.0 will not be output in incremental mode!

Typical Example of Numeric Modal:

Non-modal output

Modal output

G01 X3. Y4

G01 X3. Y4.

G01 X3. Y8.

Y8.

G01 X4. Y8.

X4.

AXIS Modals

This is a modal type 2. It is used for numeric input with axis motion. X, Y and Z entries use this type of modal. It is handled the same way as a numeric modal except it keeps track of the absolute position of the axis regardless of incremental or absolute mode. Use this modal for all axis type entries in the \$pp file.

Example:

`X#5.4#6.2;M#2=X`

`Y#5.4#6.2;M#2=Y`

`Z#5.4#6.2;M#2=Z`

These lines determine that the X, Y, and Z axis calls are a modal type 2 because they are each axes called in the postprocessor. They would output a letter (X, Y, or Z) and either a 5.4 format in inch mode, or 6.2 format in metric mode each time one of them are called into play.

Reserved Words

The postprocessor contains many hard coded keywords that usually are not changed. These words are referenced by the program and have specific behavior which is described in this section.

\$ARC1.1

\$LINE1.1

\$POINT1.1

\$TEXT1.1

These keywords are used to indicate geometry definitions in the \$PP file. The 1 left of the '.' stands for group 1, the 1 to the right of the '.' indicates the first line to be output to the NC file. Used in conjunction with text statements and the ;G switch, this allows for different types of output for geometry. See the section on group definitions for more information.

There may be more than one of each of these depending on the output necessary when each of these groups are called. For instance there may be a \$ARC1.1 and a \$ARC2.1 depending on the behavior necessary for the machine.

\$BLANKLINES

This is a post entry that allows for blank lines to be produced. Normally, all blank lines are suppressed by the postprocessor.

Example:

```
=$BLANKLINES
```

Will allow the post to output blank lines.

\$COMMAISON

```
;; =$COMMAISON
```

Normally the postprocessor will remove commas from text statements as per the APT standard.

Example:

```
CYCLE1/10.0,R3,P7
```

would generate **G88X10.R3P7** in the NC file.

However, some machine tools need commas in the NC file to separate parameters.

Example:

```
=$COMMASON
```

```
CYCLE1/10.0,R3,P7
```

would generate G88X10.,R3,P7 in the NC file.

\$FIRSTINDEXOFF

```
;; =$FIRSTINDEXOFF
```

This line (which is commented out in GENERIC) controls the generation of the first index line. This is a line which is developed from the machine tool home position (usually 0,0,0) to the startpoint of the first geometric element.

Example:

```
=$FIRSTINDEXOFF
```

Would suppress the generation of the first index line.

\$HEADER.1

This is a word in the \$PP file that indicates a code to be processed on startup of CPOSTII. This code will NOT generate NC code, it is specifically designed to establish the default state of the machine tool. The '1' following the word is used to indicate multiple header codes. The software will process \$HEADER.1, \$HEADER.2, and so on.

\$MATCHTOL

```
0.0001=$MATCHTOL
```

This keyword controls the index line match tolerance between geometric entities. If two consecutive entities are greater than 0.0001 unit apart, an index line will be developed. Note that this constant also applies to index line development during validation.

\$OUTTOL

```
8=$OUTTOL
```

This line sets the default resolution used during post processing and computation of block transforms to eight decimal places. In order to obtain higher accuracy this value can be set as high as 16 decimal places, but at the expense of slower operation.

ABS/INCR

```
ABS=$HEADER.3
```

These keywords are used to switch the output mode of the postprocessor to absolute or incremental. In a \$HEADER line these keywords establish the default state of the machine tool (i.e. GENERIC assumes the machine is in absolute mode when powered on).

Example:

```
INCR=$HEADER.3
```


Would cause the postprocessor to startup in incremental mode.

These keywords can also be entered into the AutoCAD drawing as text statements to allow switching between incremental and absolute mode during a NC program.

ARCDROP

This represents the number of the lines to drop if an arc is equal to 90 degrees. It can be used on multi-line arc definitions to drop the remainder of the definition.

Example:

```
SEQNO#XC#YC... =$ARC1.1
```

```
SEQNO#SWEEP#... =$ARC1.2
```

```
SEQNO#X#... =$ARC1.3
```

```
;I#2=ARCDROP
```

Will drop the \$ARC1.2 and \$ARC1.3 from the output if the arc is equal to 90 degrees.

ARCTOLINE

This is the smallest arc (chordal) that can be represented by the machine tool. If the delta X or the delta Y from the start point to the end point of the arc is less than this value, the arc is converted to a straight line.

Example:

```
;I#0.001=ARCTOLINE
```

Will turn all arcs whose delta X or delta Y is less than 0.001 into lines.

ARCTOLINE is for quadrant arc posts only

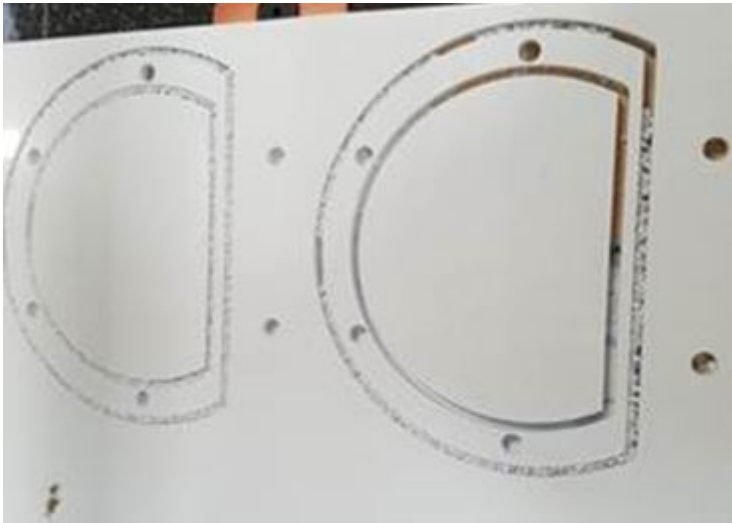
ARCTOLINE looks for a change in X Y and Z with any 2 being identical the ARC will be turned into a line.

If you use this in a regular non-quadrant arc post:

In this example geoshape create this object as 3 arcs.

In this case the Y and Z components DID NOT change so it was turned into a line.

Had Quad Arc kicked in we would not have this exact match in Y and Z so the arc would have been put out.



ARCTOLINE should be in EVERY quad arc post.

The logic behind Arctoline:

```

/* $NOTE$ - the logic for ARCTOLINE depends upon quadrant arcs! */
if ((tol = ppgetf(_T("ARCTOLINE"))) > 0.0) {
    if (scaleon) tol *= fabs(scalef[0]);
    k = 0;
    xc2d = fabs(endx - x);
    yc2d = fabs(endy - y);
    zc2d = fabs(endz - z);
    if (xc2d >= tol) ++k;
    if (yc2d >= tol) ++k;
    if (zc2d >= tol) ++k;
    if (k < 1) goto CONT; /* all dropped out */
    if (k < 2) {
        ppoutlin(f, loc_cur, loc_ppmode, loc_cur[3], loc_cur[4],
loc_cur[5], endx, endy, endz);
        goto CONT;
    }
}

if (dir == -1) {
    /* $M$ 980729 CArps moved this AFTER ARCTOLINE*/
    ppputf(_T("DIR0"), 0.0);
    ppputf(_T("DIR1"), -1.0);
    if (lookup(_T("CIRCUL/CLW")) != NULL) {
        pform(temps, _T("CIRCUL/CLW"), 0.0);
    }
}
else if (dir == 1) {
    ppputf(_T("DIR0"), 1.0);
    ppputf(_T("DIR1"), 1.0);
    if (lookup(_T("CIRCUL/CCLW")) != NULL) {
        pform(temps, _T("CIRCUL/CCLW"), 0.0);
    }
}

```

BEGIN

`%=BEGIN`

This line defines the string to be output at the beginning of the NC file. This string typically defines the start of a program to the machine tool controller.

CIRCINT

If not 0, circular interpolation is turned on and all arcs are converted to straight line segments.

Example:

`CIRCINT/1.0`

Or

`;I#1.0=CIRCINT`

Will turn on interpolation for all arcs.

CIRCLEN

This is related to arc interpolation. If the arc length is greater than this value, circular interpolation is turned on for this arc only.

Example:

`;I#99.999=CIRCLEN`

Will turn on arc interpolation for any arc whose arc length greater than 99.999.

CIRCUL/CLW and CIRCUL/CCLW

These keywords are reserved because they are automatically executed when an arc is processed. If a clockwise arc is processed by the postprocessor, "CIRCUL/CLW" definition in the \$PP file will be evaluated. If a counterclockwise arc is processed, the "CIRCUL/CCLW" definition will be evaluated. This results in all switches being processed and allows disable/enable of codes depending on arc direction. It also allows the ;G switch which allows totally different group definitions that automatically switch based on arc direction.

CIRCRAD

This is related to arc interpolation. If the arc radius is greater than this value, circular interpolation is turned on for this arc only.

Example:

`;I#9.999=CIRCRAD`

Will turn on arc interpolation for any arc whose radius is greater than 9.999.

END

`%=END`

This line defines the string to be output at the end of the NC file. This string typically defines the end of a program to the machine tool controller.

EOB

`^013^010=EOB`

When the .NC file is converted to the .OUT file, this line determines what the '\$' character at the end of each line will be translated to. The default in GENERIC is to output a carriage-return line-feed at the end of each block of NC code.

Example:

`$=EOB`

Would cause all lines or blocks of NC code to be separated only by the '\$' character in the .OUT file. No carriage-returns or line-feeds would be present in the .OUT file.

ERRORC

This is a post entry that will turn on error correction for incremental machines. The error is accumulated and added to the next linear motion. If set to 1, only lines are error corrected. If set to 2, both lines and arcs are error corrected.

Example:

`; I#2=ERRORC`

Will turn on error correction for both lines and arcs.

FIRST

`;; =FIRST`

This line defines the string to be output at the beginning of the .OUT or final output file. This is only used when there is a non-printable string needed at the start of a program.

Example:

`^007=FIRST`

Would place an ASCII 7 or control-G at the start of the .OUT file.

FLUSH

When this text statement is processed by the postprocessor, all pending text statements that are in buffers will be flushed out to the NC file. This is accomplished by the post executing the \$TEXT definition. Normally text statements are flushed out on geometry, but sometimes it is easier to collect text statements and then FLUSH them out.

FORMAT

`ASCII=FORMAT`

This line controls the output mode of the postprocessor. GENERIC is set up to output ASCII code to the final output file (the .OUT file).

Example:

```
EIA=FORMAT
```

Would cause the postprocessor to generate EIA code to the final output file. Note that the .NC file is still ASCII and can be hand edited and then converted to EIA using menu selection 2 of the CPOSTII main menu.

\$HELIX

A 3d polyline on layer NC_HELIX will be considered a helix if the \$PP entry \$HELIX is set.

Example:

```
= $HELIX
```

Will generate code for a helix if a polyline is on layer NC_HELIX. The start and endpoint are the same as the polyline. The radius and direction are computed from the polyline.

HELIXD

Example:

```
=HELIXD
```

Will generate for a helix with the resulting output change in degrees per unit.

HOME

```
HOME/0.0,0.0,0.0=$HEADER.2
```

The postprocessor needs to know where the machine tool is positioned at the start of the program. This line initializes the machine tool position at startup to X0.0Y0.0Z0.0.

Example:

```
HOME/-20.0,-19.0,10.0=$HEADER.2
```

Would set the postprocessor so that the initial position of the machine tool is at X-20.0Y-10.0Z10.0.

INDEXGROUP

This is the group number that will be switched to when a line on a layer named NC_INDEX is processed.

The lines layer only need start with NC_INDEX to be counted as an index line. NC_INDEX_FR, NC_INDEX, NC_INDEX_TO are all valid index layer names.

Example:

```
;I#4=INDEXGROUP
```

Will cause the postprocessor to switch to group number 4 when an index line is encountered. The post will then switch back to the previous group after the line is processed.

INTOL

This is the value used to compute the arc deviation for arc interpolation. The smaller the value, the more line segments and accuracy is obtained.

Example:

```
;I#0.001=INTOL (good for inch machines)
;I#0.01=INTOL (good for metric machines)
```

JOBID

```
:=JOBID
```

This keyword will result in a line of NC code at the top of the program with a job identification number. If this line is present in the \$PP, the postprocessor will output a JOBID typically with a number following the characters specified in this line.

LAST

```
;; =LAST
```

This line defines the string to be output at the end of the .OUT or final output file. This is only used when there is a non-printable string needed at the end of a program.

Example:

```
^007=LAST
```

Would place an ASCII 7 or control-G at the end of the .OUT file.

LINEAR/RAPID

```
RAPID=$HEADER.4
```

This line represents the default state of the machine tool in terms of cutting verses non-cutting motion. GENERIC assumes that the machine tool is in non-cutting or RAPID mode. You should note that when this code is evaluated at startup, it disables the LINEAR code with the ;D switch.

Example:

```
LINEAR=$HEADER.4
```

Would cause the post to assume that the machine tool starts up in cut or LINEAR mode.

These keywords can be entered as text statements to switch between LINEAR and RAPID mode during an NC program.

METRIC

This keyword will cause the postprocessor to use the secondary or metric formats on all quantities.

ORIGIN

```
ORIGIN/0.0,0.0,0.0=$HEADER.1
```

This is the first code to be executed in GENERIC.\$PP and sets the origin of coordinates. These three values are subtracted from each X,Y, and Z value when it is output to the NC file.

Example:

```
ORIGIN/-40.0,-30.0,0.0=$HEADER.1
```

Would result in 40 being added to each X value and 30 added to each Y value on output to the NC file.

POFF

This is a post entry that defines an offset for coordinate output. This offset is in addition to the OFFSET \$PP entry.

Example:

```
ORIGIN/1,1,0
```

```
POFF/2,2,0
```

Will result in a net offset for coordinate output of 3,3,0.

SEQINC

This word specifies the increment amount that SEQNO is increased by. The default in GENERIC is 1, but can be anything.

Example:

```
;I#10=SEQINC
```

Would set sequence numbers to increment by 10.

SEQNO

This keyword is used to indicate an incrementing variable used to generate sequence numbers. Note that sequences numbers can be reset by a text statement in the drawing.

Example:

```
"SEQNO/1000" (text statement in a drawing)
```

Would set the current sequence number to 1000.

ZAXIS/Z

These keywords are identical and provide a way for the user to enter Z coordinate values with text statements. If the current mode of the postprocessor is absolute, then all Z text is considered to be absolute Z values. However, if the post is in incremental mode, then all Z values entered via text statements will be considered relative or incremental values.

Fine Tuning

The .SPP file shipped with the software has been configured according to the controller's manual. Any optional codes or canned cycles should be added to the .SPP file to ensure proper code production. The Router-CIM Postprocessor will show an error message stating that "VOCABULARY WORD NOT IN .SPP FILE" if it cannot find a specific code. There is always a certain amount of "fine tuning" involved in getting the system to produce code that looks the way you want it to look.

If changes need to be made, they should be made in the [controller].SPP file. This file determines the final placement of codes and vocabulary for the system. Use the DOS EDLIN or a text editor in the "non-document" mode to make the changes.

Switch Parameters

The following lists all of the available switches and explains their use.

;A

An entry using this switch will have its NC code placed in the AFTER or PA buffer so when the motion block is processed, the code will be placed after the motion information.

Input **G04 ;A**

Result X2000Y2000G04

;B

An entry using this switch will have its NC code placed in the BEFORE or PB buffer so when the motion block is processed, the code will be placed before the motion information.

Input **G04 ;B**

Result G04X2000Y2000

;C

An entry using this switch will have its NC code placed in the CURRENT or PC buffer so when the entry is processed, the NC code will be immediately output to the NC file. In terms of group definitions, the \$TEXT group in the .SPP file will be evaluated and output to the NC file.

Input **G04 ;C**

Result G04

;D

An entry using this switch will disable other .SPP entries so they do not generate output. The parameters to this switch are other .SPP entries which are to be disabled. Note that all codes are initially defaulted "ENABLED".

This switch can be used in conjunction with the "E" or "ENABLE" switch to handle codes that enable/disable each other.

For example, "**G00;D#LINEAR;E#RAPID=RAPID**" and "G01;D#RAPID;E#LINEAR=LINEAR" will define "G00" to be output when the text "RAPID" is processed and "G01" to be output when "LINEAR" is processed. When "RAPID" is processed it will disable the \$PP entry "LINEAR" and enable itself. This provides the ability to output G00 or G01 in a motion block definition as well as the ability to disable/enable other \$PP entries.

;E

An entry using this switch will enable other \$PP entries so they do generate output. The parameters to the switch are the \$PP entries to be enabled. This switch is used in conjunction with the "D" switch to output one code or the other.

;F

An entry using this switch will not use global formatting switches for number formatting. The required options will be taken from the 'F' switch.

For example, "**N#4#4;F#ZL=SEQNO**" will create fixed left sequence numbers with leading zeros ("N0001" as opposed to "N1").

;G

This switch controls the current group number for \$PP output. Use this to allow for totally different blocks of NC code to be generated. The default group is set to 1 on startup of the post.

;I

An entry using this switch will be initialized to the value contained in the parameter.

Example: "**N#3.0;I#1=SEQNO**" initializes sequence numbers to 1.

Note that all values without a ;I switch will be initialized to 0.

;M

This switch is used to control the definition of modal codes to the postprocessor.

;N

This is a switch that controls the modal behavior of the current line of NC code. If a line completely modals out, the lines output is suppressed. SEQNO is always considered to be blank so that lines of code containing only SEQNO will be suppressed.

This switch in ;N#0 form will cause the code to always be considered blank for modal purposes (just like SEQNO).

In the ;N#1 form, this switch will cause the entire line to be suppressed if this code modals out. This can be used to suppress tool change lines if the tool modals out.

Example:

T#4#4;M#0;N#1=TOOLNO

Will suppress the current line of NC code if the TOOLNO has modaled out.

;P

This is a switch that will put or assign values to other \$pp entries. The first number after the switch controls the type of assignment. The number 0 indicates an absolute assignment, the number 1 indicates an incremental assignment. After the first number you can repeat pairs of \$PP names and numbers to assigned to them.

Example 1:

```
;P#0#SEQNO#100#SEQINC#10
```

Will place the number 100 into the post entry SEQNO and 10 into SEQINC.

Example 2:

```
;P#1#SEQNO#100
```

Will increment SEQNO by 100.

An entry can also be assigned a value by using a variable instead of a fixed value.

Example :

```
;P#0#SEQNO#3
```

Will place a 3 into SEQNO.

This could now be accomplished with the following statements.

```
;I#3.0=SI
```

```
;P#0#SEQNO#SI
```

With the added benefit that you could set SI in the drawing and the post would output the correct value.

;Q

This switch is used to control the generation of quadrant arcs by the post. If this switch appears on either of the entries "CIRCUL/CLW" or "CIRCUL/CCLW, then all arcs will be output broken along quadrant boundaries.

Example:

```
G02;D#CIRCUL/CCLW;E#CIRCUL/CLW;M#0#LINEAR;N#0;Q=CIRCUL/CLW
```

This line would force Clockwise arcs to be broken up into quadrants.

;S

This switch is used to scale the value before output to the NC file. The most common use for this switch is in lathe applications that require one axis to be specified as diameter. "Z#3.4;S#2.0=X" would cause all X axis output to be output with a 'Z' letter code and multiplied by 2.0.

;V

This switch is used to retrieve a numeric value from one code to be used in another.

Example:

```
R#3.3;V#X=MYX
```

This line defines entry "MYX" to get its value from the current value stored in post entry "X". This switch can also be used to define X,Y, or Z axis output that is not modal. A typical example would be a controller that needs X and Y for a drilling cycle at the current point. If X, Y, and Z are defined to be modal, a different X can be defined that is not modal.

Example:

```
X#3.4;M#1=XMOD
```

```
X#3.4;V#X=XNON
```

;Y

This is a switch that functions exactly like the 'D' disable switch but 'Y' will not reset the modal buffers. The 'D' switch currently disables a code and resets its modal buffers.

Example:

```
;Y#LINEAR;E#RAPID=RAPID
```

Will cause RAPID to be enabled and LINEAR to be disabled, but LINEAR will not have its modal buffer reset.

;Z

This is a switch that allows a string to be appended to the end of a code.

Example:

```
(X#3.4#3.4;Z#)=X
```

Assume X is currently 2.0 and this will generate (X2.0) as output.

General Format of the ;P Switch

```
;P#TYPE#VARIABLE#VALUE
```

Where type indicates the type of operation to be performed:

0	=	Place VALUE into VARIABLE
1	+	Add VALUE to VARIABLE
2	-	Subtract VALUE from VARIABLE

3	*	Multiply VARIABLE by VALUE
4	/	Divide VARIABLE by VALUE
5	Sin	Set VARIABLE to sin(VALUE)
6	Cos	Set VARIABLE to cos(VALUE)
7	Atan	Set VARIABLE to atan(VALUE)
8	Mod	Set VARIABLE to mod(VALUE)

NOTE: Due to syntax restrictions only one TYPE of ;P switch can be used per \$PP entry.

"F" Switch Parameters

The "F" switch requires one or more parameters to determine what type of numeric formatting to perform. The parameter(s) is delimited by a "#" followed by each required single character parameter. An example:

```
N#4#4;F#ZL=SEQNO
```

The Z indicates 'leading zeros' and the L indicates 'fixed left'.

An alternate method of numeric formatting is provided using header codes. Each individual header code can contain one parameter. The parameter would be the character string representing the desired selection. When using header codes ALL numbers are affected. However, if an individual item has the "F" switch, the "F" switch will over ride the header code. An example header code is:

```
UNSIGNED=$HEADER.6
```

This header code will make ALL numeric values UNSIGNED (no + or - signs).

The following table describes the available parameters for the "F" switch.

PARAMETER	HEADER CODE	DESCRIPTION
-----------	-------------	-------------

-	SIGNREV	An entry with this switch will output its numeric value with the sign reversed to NC code.
+	PLUS/ON	An entry with this switch will output positive numeric values with a '+' at beginning of the number.
D	DECIMAL/OFF	An entry with this switch will output its numeric value with the decimal point removed.
O	ROUND/OFF	An entry with this switch will output its numeric value with no rounding applied.
J	LEFTJUST	An entry with this switch will output its numeric value left justified.
L	FIXED/LEFT	An entry with this switch will output its numeric value with the number to the left of the decimal point a fixed number of places.
N	N/A	An entry with this switch will output its numeric value with a single leading zero (0) added to it.
R	FIXED/RIGHT	An entry with this switch will output its numeric value with the number to the right of the decimal point a fixed number of places.
S	N/A	An entry with this switch will output its numeric value with a single trailing zero (0) added to it.
T	TRAILING/ZEROS	An entry with this switch will output its numeric value with trailing zeros added to it. Note that FIXED/RIGHT must also be on for this option to work.
Z	LEADING/ZEROS	An entry with this switch will outputs its numeric value with leading zeros added to it. Note that FIXED/LEFT must also be on for this option to work.

U	UNSIGNED	A code using this switch will have its sign stripped off and be output as an unsigned value.
----------	----------	--

Geometric Entries

A variety of geometric information is computed by the postprocessor. This geometric information can be used as \$PP entries. This provides for various methods of representing geometric information for different types of controllers.

The following describes the different types of geometric entries allowed in the \$PP file:

ANGLE	Angle from start point to end point in degrees.
DIR0	Direction of arc, 0 if counterclockwise 1 if clockwise.
DIR1	Direction of arc, -1 if counterclockwise 1 if clockwise.
EA	End angle of arc in degrees.
LENGTH	Length of line or arc.
RADIUS	Radius of arc.
FRADIUS	This is the radius of the current arc/circle but it will be negative if the sweep is greater than 180 degrees.
SA	Start angle of arc in degrees.
SWEEP	Sweep of arc in degrees, positive if counterclockwise, negative if clockwise.
X	Current cutter position in X. Absolute or incremental depending on mode.
X1A	Absolute X coordinate of start point of arc, line, point.
X2A	Absolute X coordinate of end point of arc, line.
X2I	Incremental X coordinate of end point of arc, line.

XA	Current absolute cutter position in X.
XC	X coordinate of center of arc. Absolute or incremental depending on mode.
XCA	Absolute X coordinate of center of arc.
XCI	Incremental X coordinate of center of arc.
XI	Current incremental cutter position in X.
Y	Current cutter position in Y. Absolute or incremental depending on mode.
Y1A	Absolute Y coordinate of start point of arc, line, point.
Y2A	Absolute Y coordinate of end point of arc, line.
Y2I	Incremental Y coordinate of end point of arc, line.
YA	Current absolute cutter position in Y.
YC	Y coordinate of center of arc. Absolute or incremental depending on mode.
YCA	Absolute Y coordinate of center of arc.
YCI	Incremental Y coordinate of center of arc.
YI	Current incremental cutter position in Y.
Z	Current cutter position in Z. Absolute or incremental depending on mode.
ZA	Current absolute cutter position in Z.
ZI	Current incremental cutter position in Z.

Customization

Once you feel comfortable with the basic workings of the system, you can begin to customize it to fit your specific needs. Router-CIM has provided you with the tools to accomplish this, primarily through the use of the \$pp file, the tasks function, and the Sequencer file.

You must be familiar with a text editor that is capable of a non-document mode, such as those mentioned at the beginning of this book, in order to proceed. You should additionally have a complete backup of your system before modifying any of its files. The author assumes no responsibility for any damage or lost time or materials resulting from your modification of a your post processing system.

A task is a series of statements to Router-CIM system which either gives a set of responses or allow for user input which produces AutoCAD text statements.

For instance, let's assume that there are several codes which come at the start of each one of your programs.

G17G70G90

M27M47M26G92X0Y0

Since each one of these codes is accessed by a vocabulary word in the .SPP file, you could use AutoCAD text command to type in the seven words to produce these codes. However, by fully utilizing the .SPP file and a Router-CIM task, we can reduce our work to a single operation.

In the .SPP file use Notepad or a text editor in non-document mode to make the following additions.

G17G70G90 ; C=SC1

M27M47M26G92X0Y0 ; C=SC2

The ";C" means the code(s) will be produced on a line of its own. "SC1" and "SC2" have been randomly chosen to stand for "Start Code 1" and "Start Code 2." Thus if the text "SC1" appears in a drawing it will cause G17G70G90 to appear in the code.

Next, create a new task. Type in the following two lines:

SC1

SC2

Name the task "START".

When you run the task "START" it will automatically write the text "SC1" followed by "SC2".

This same technique can be used to access tool changes, drill cycles, end of program routines and any other codes or group of codes. For our purposes this might take the form of a parameter to indicate spindle speed or feedrate, or both. We might create a situation like the following:

M21M08M03S ; C=SPCLON

This "SPCLON" would turn on the spindle in a clock-wise direction, start the coolant and supply the spindle speed in the form of the "S" parameter which might be different for different jobs. Therefore, we would want to put it in each time, but still within the task.

SPCLON 1200

The command line would read:

Text: SPCLON,

We would add "1200" to produce a text statement "SPCLON,1200" which would eventually become "M21M08M03S1200" after processing through Router-CIM and the postprocessor. If you needed to add a FEDRAT as well, you would add "1200F35" which would eventually become "M21M08M03S1200F35."

Video Jet Print Head

There are 2 basic way to interface the Video Jet print head in the Router-CIM Automation Suite.

The First way is the place text statements directly on the part drawing. These text statements must be on layer LABELS. All text should be **capitalized**. No lower case letters should be used. When the text is found in the drawing Automation Suite will convert them to proper printing instructions during tool path generation.

These text statements would be converted to layer LABELS1 for font 1. You may also use layer LABELS2 and LABELS3 for font 2 and font 3.

The best use is to place text on layer LABELS1.

The second way to interface to the printer is through the use of part record label fields. There are 8 user controllable label fields available from the interface or directly from imported data (excel .XLS files or .CSV files).

Basic support fields 1-4, are 24 characters each. Fields 2,3,4 can be combined to print in 1 stroke with the use of the ***VJMLine*** variable

Enhanced support for label fields 5-8

Field 5 (font 3)

fields 6 7 8 support multi line (font 2).

Fields are limited to 60 chars each if multi line is turned on due to maximum NC code line length of 255 chars. If single line text is selected then each of the fields can be 99 chars long.

The basic functionality in Router-CIM is to match the height and length of the text with a line of the proper length to synchronize the machine tool movement to allow the proper time for the videojet head to print all the characters on the part. Router-CIM will evaluate the text characters and create a 'tool path' that is as long as it needs to be in order to print all characters on the part.

User controllable variables

VJMLine

This variable will turn on support for 3 fonts.

Font #1 is tied to Label field 1.

Font #2 will be tied to Label field 2-3-4.

The 3rd font is tied to label field 5. Fields 6,7,8 are tied to font 2.

**Note font 2 must be a multi line font.*

VJTXHTGT

This allows user control over the TEXT height that appears in the drawing from the label fields. This will allow the user to match the actual text height they are printing with what the videojet printer is set to.

Note: text height in AutoCAD also controls interline spacing.

Acceptable ranges for this variable are between 0.1875 and 0.375 in inch mode and 4.7mm to 9.5mm in metric mode.

The default value for this variable is 0.1875

VJXTSCALE

This variable allows users to control the scale of the text to be printed. The printer may be set to a faster line speed than the knowledge / machine is set to. This will allow a font that appears compressed but it will print faster due to the smaller machine motion. A scale factor of 1 is the default, and matches the ACAD font to the VideoJet font.

For example: setting the line speed to 750 IPM (Machine @ 500 IPM) and placing a factor of 0.75 will create 25% compression in the appearance of the text. The Text would appear slightly squashed but still very readable.

Acceptable ranges for this variable are between 0.75 and 1.5.

Router-CIM Layer to Knowledge for Video Jet

There is a special knowledge that loads with the postprocessor for all VideoJet machines. It is named VJTXT and controls the conversion of text statements into NC machine instructions. Video Jet post-processors will add the extra cycle, tool number and tasks for the Video Jet Marking head. For Automation, this knowledge must be in the knowledge drawing for the current job. The post will be set up to output Z0 only. The responsibility of setting the video jet head is stored in the tool offsets because only 1 height is ever used.

You can place data in label fields 1-8 in Automation and Router-CIM will generate the correct tool paths and code to make the Video Jet head work if there is a knowledge called VJTXT in the drawing. No layer to knowledge association is necessary for this to function.

Additionally, you may place text on layer "LABELS" and if the VJTXT knowledge exists in the drawing when automation runs the part, it will also create tool paths and code for the Video Jet head.

Here is a sample NC Code program and some explanation of the Video Jet features.

```
N5(ROUTER-BIT .5 DIA.)
N6G28G91Z0M05
N7G90G53C0.
N8G53A90.
N9G00G17G54X-22.6733Y-1.9043
N10G00G43H16Z1.25M16(M16 LOWERS THE HEAD IN PLAY M17 FIRES THE PRINT HEAD)
N11G01Z.75F175.
N12X-18.4233F350.M17(TXT:Hull 662D Bay C2 HGT:0.25 FNT:1)
N13G00Z1.25
```

```
N14G00X-22.6733Y-2.4043
N15Z.85
N16G01Z.75F175.
N17X-19.4233F350.M17(TXT:Layer 2 of 23 HGT:0.25 FNT:1)
N18G00Z1.25
N19G00X-22.6733Y1.5957
N20Z.85
N21G01Z.75F175.
N22X-19.4233F350.M17(TXT:Layer 2 of 23 HGT:0.25 FNT:1)
N23G00Z1.25
N24(ROUTER-BIT .5 DIA.)
N25G28G91Z0M05
```

In the above example the Line 10 uses the M16 to activate and lower the print head
Lines 12,17,22 each have the required M17 to send that buffered string to the print head.
The X movement corresponds to the length of the printer movement required to spray the text on the part.
The TXT: statement is the text to be printed.
The HGT: statement is the height of the text in the drawing.
The FNT: statement is the font number used to print.

Video Jet Technical Notes

Technical data about multi line support. The items described here are not a concern for users and only describe the inner workings of the technology.

```
CAPITOL
12345678
KOMO
```

Even though theses are 3 separate records of data they must be passed to the printer in the following order. This tool was mainly designed for use with the printer screen not on the fly edits via a serial port.

C1KA2OP3MI4OT5 O6 L7 8 The first char from each line then the second and so on with spaces padded for odd line lengths.

Font control

Font selection is limited to size, Proportion and quality. There are not many choices of actual fonts. See the Video Jet manuals for referencing font codes for Komo Production Manager.

Video Jet Frequently Asked Questions

Listed below are some common questions and answers about the Video Jet Printer Head.

Q: Can the system be setup to reduce the potential for a crash? As an example, I customized our current NC Post Processor so that it would not generate code if a router tool cut depth exceeded the panel depth by more than 1/16".

A: Yes, the head distance away from the top surface of the part controls the text spray size(height). This can be set in the knowledge of the cut. *Caution must be exercised as you get closer the surface of the material on order to prevent crashes.*

Q: How is the information supplied to the 4-label fields?

A: CSV,Excel import, or data entry directly into the parts screen are the possible methods to populate a part record with label data

Q: Can the text be printed smaller than the .1875 - .375wide characters as noted? At this size a string of 20 characters would be 5"wide.

A: Yes, the head distance away from the top surface of the part controls the text spray size(height). *Caution must be exercised as you get closer the surface of the material in order to prevent crashes.*

Q: Is inserting text on layer LABELS the only way to drive the Video Jet Head?

A: That is only 1 of the 2 methods possible. Imported data as discussed above might be the better way to insert label data.

Q: Where does the information come from for the inkjet label (part drawing, etc.)

A: Each part record has up to 4 label fields that are available to be printed on the center of the part but the user must consider the text size and text length in there printing. Each char in a printed message is approx .25-.375 in length. In addition the user may place text on a layer called LABELS and this will also be printed on the part.

Q: What dictates the orientation of the label (0 or 90 degrees)

A: All text will be printed with the rotation that nesting assigns to the part automatically.

Q: Speed of the inkjet labeler/printer

A: 450 IPM is the best speed to print so far.

Q: I only get a few characters printed and then a blob of ink at the end.

A: This seems to be caused by either changing the feedrate so that it is too fast for the videojet to print the characters in the length of the move allowed, or by changing the text parameters in the NC file after the code has been produced.

Komo Production Manager and Video Jet support

On the setting page of the KPM you will need to turn on Video Jet support.

Com port should be set to 1 unless additional serial ports have been added to the computer.
The font codes listed below are for basic 7x9 (fast and looks best) for font 1.
Multi-line (prints 3 smaller lines at once) for font 2 and 6x7 for font 3 (not used at this time)

Once the VideoJET print head is enabled the buttons below will appear for accessing printer functions from KPM. See KPM manual for further instructions.

Inkjet M-Codes

The Typical M-Codes for the inkjet marking head are as follows:

M15 -- Rotate Head to print in X positive moves.
M14 -- Rotate to print in Y positive moves.
M16 -- Lower Inkjet Head
M17 -- Inkjet Print Signal

Solution machines are slightly different:

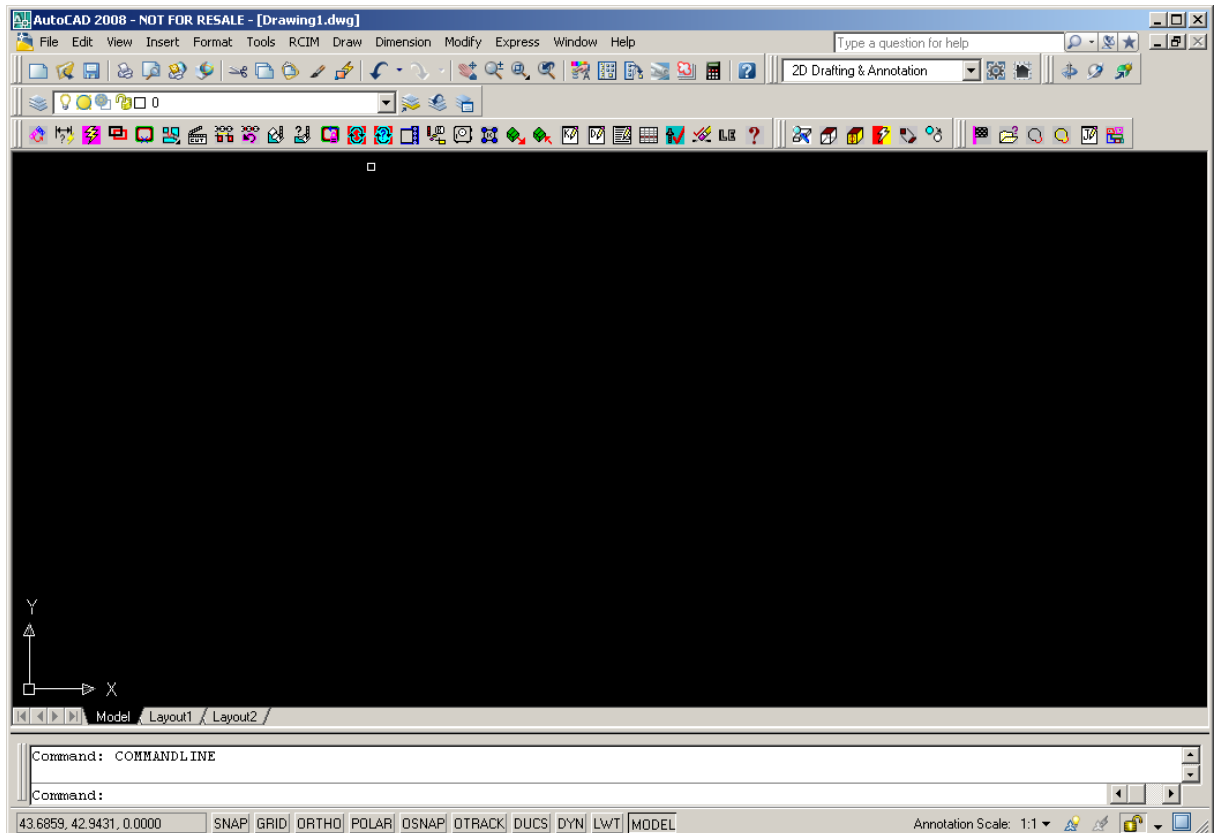
M13 -- Rotate Head to 90°
M14 -- Rotate to 0°
M16 -- Lower Inkjet Head
M17 -- Inkjet Print Signal

* If either X or Y is printing backwards, print head is installed either 90° or 180° from necessary position.
Due to the fact that print heads cannot be tested during assembly, this is a required step for the installer.

How to Backplot NC Code

There is a function built into Router-CIM to backplot NC Code files and produce on the AutoCAD screen the movements the tool made. Be aware that this will only read the NC Code file, not interpret your use of Cutter Compensation or how large a tool was, what size a hole was, etc.

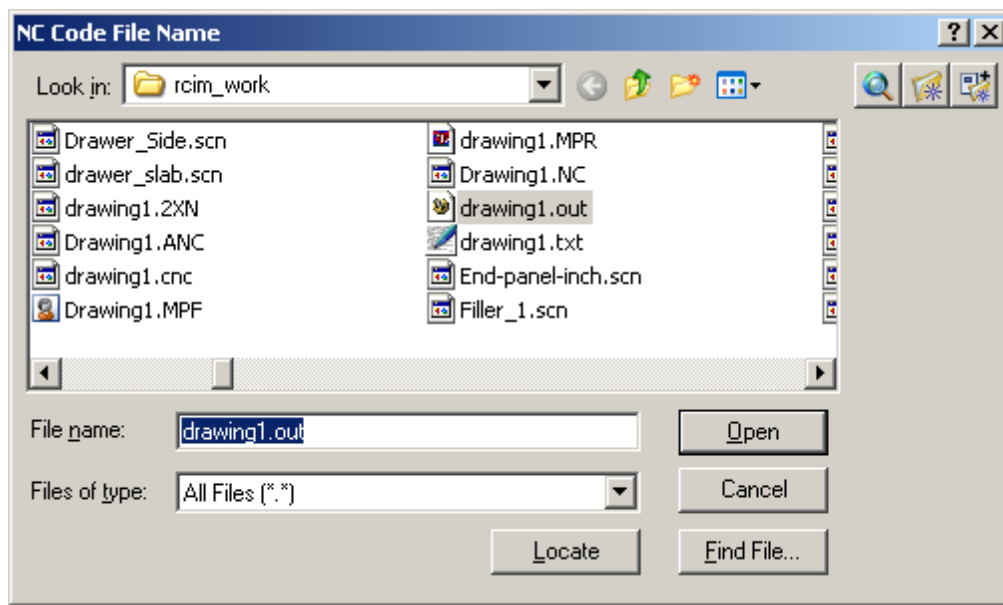
To display the tool path on the screen, first start a new, clean AutoCAD. Then start Router-CIM inside the empty drawing.



Next type BACKPLOT at them command line and press <ENTER>

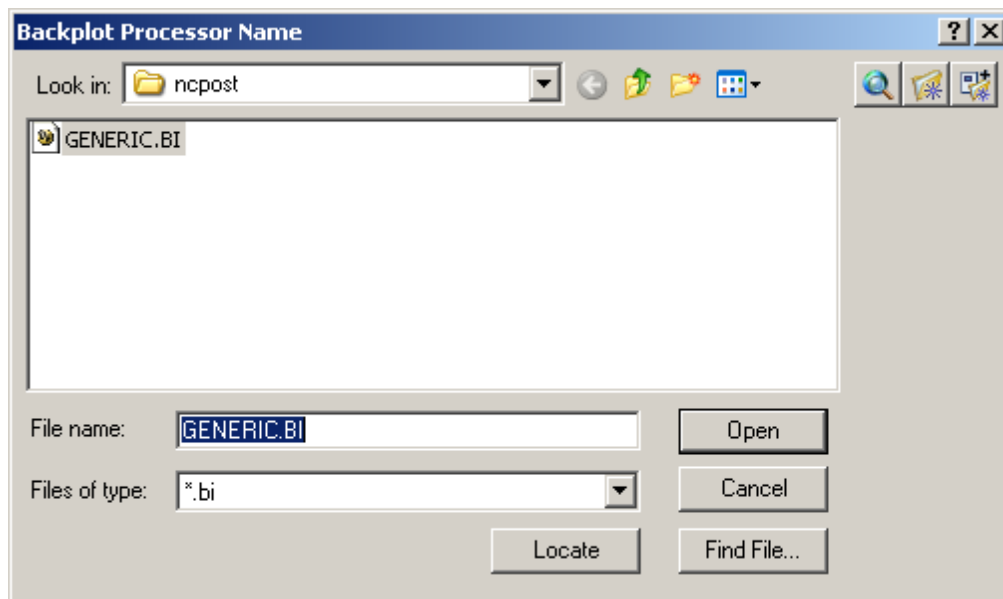


A window will appear where you can select your NC Code file. You should copy this file to the C:\Rcim_work folder as the backplot command may not work over a network path.



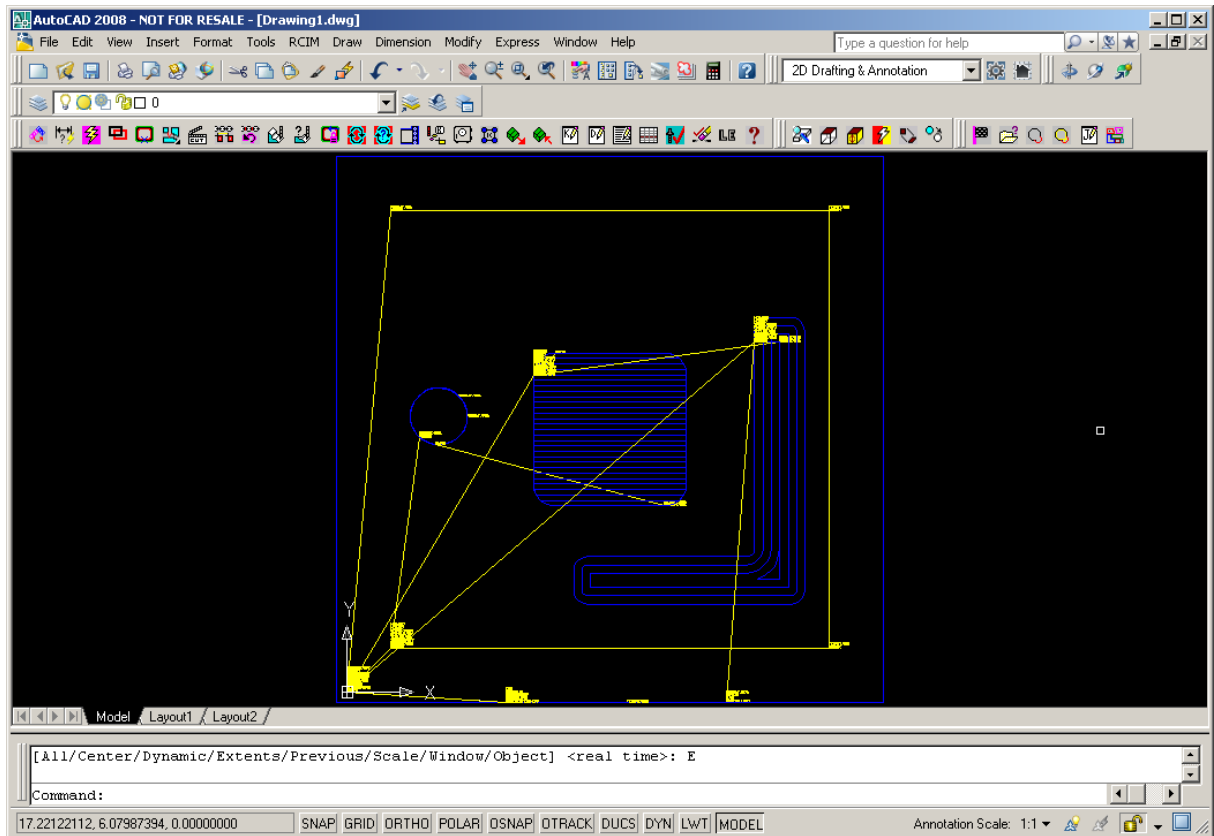
Once the file is selected, press <Open>

You will see another window looking for your backplot processor. If we have provided you with a special backplot file, it will be in the NCPOST folder and shown in this window, otherwise select the Generic.BI file.



Once the file is selected, press <Open>

The file will then be read and the code will be translated into polylines and arcs on the screen, with the Index lines and the NC Text that was processed by the Backplot processor from your NC Code file.



The real issue here is that these lines are the tools path...not your original geometry. So there are no circles for the drills to show what size they were, and the tool paths are offset or not depending on how you used Cutter Compensation in Router-CIM (or whatever software the NC Code came from). You will have to clean this drawing up quite a bit in order to re-create an actual part from it.

You can use the AutoCAD FILTER command to remove some of the bits you don't want from the drawing.

For instance, to remove all the text words from this drawing, you would first explode the drawing (as it is a block right now).

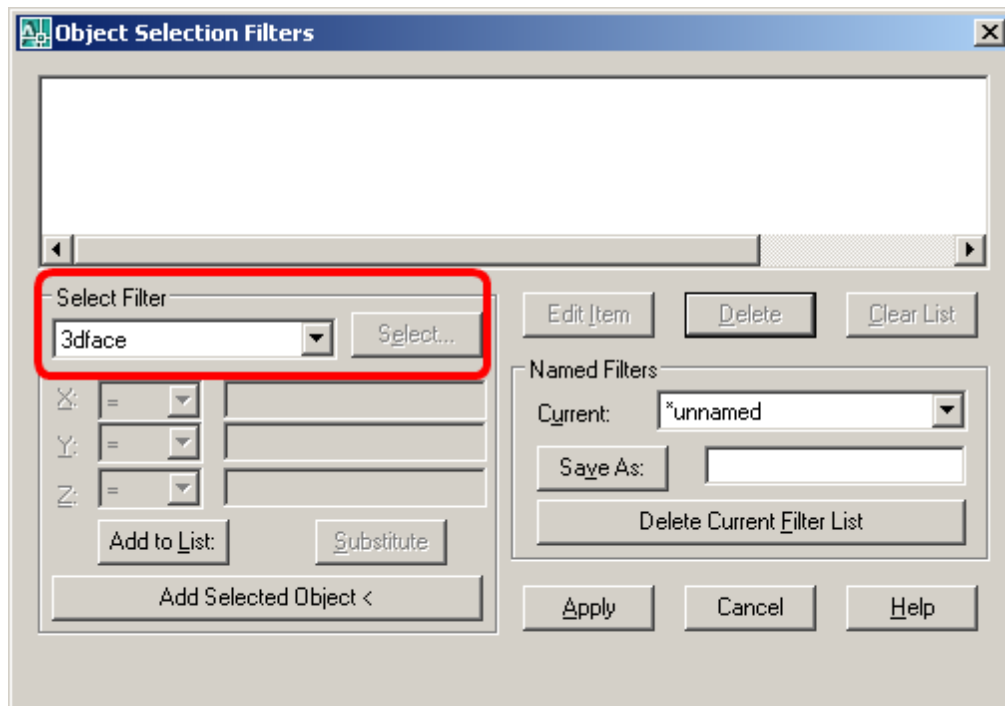
Next, start the Erase command.



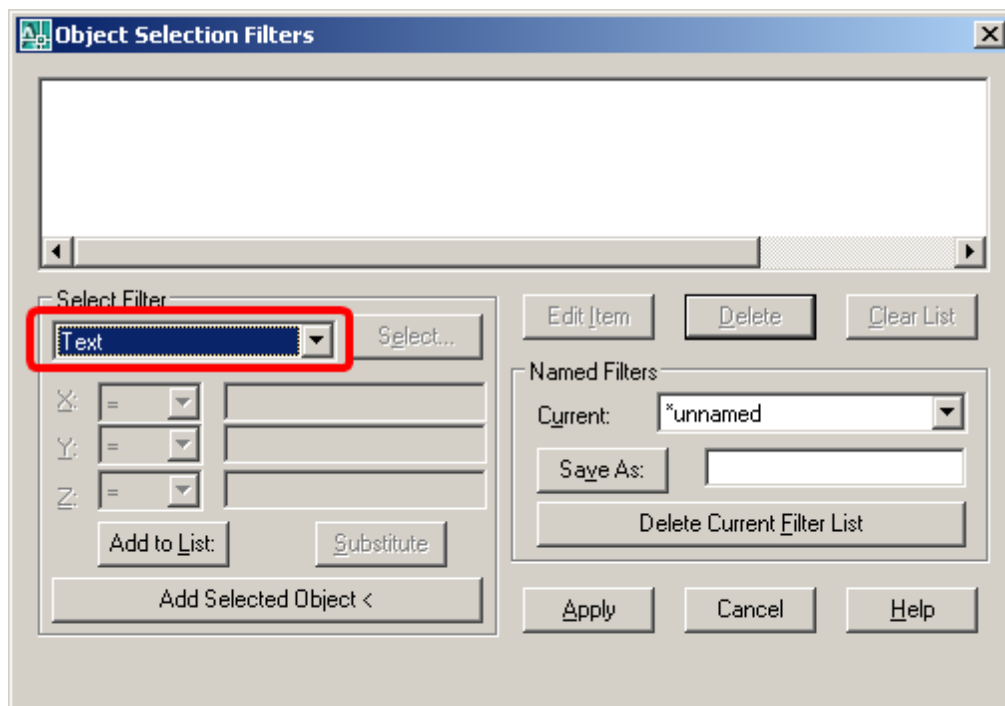
At the Select objects prompt, type 'FILTER



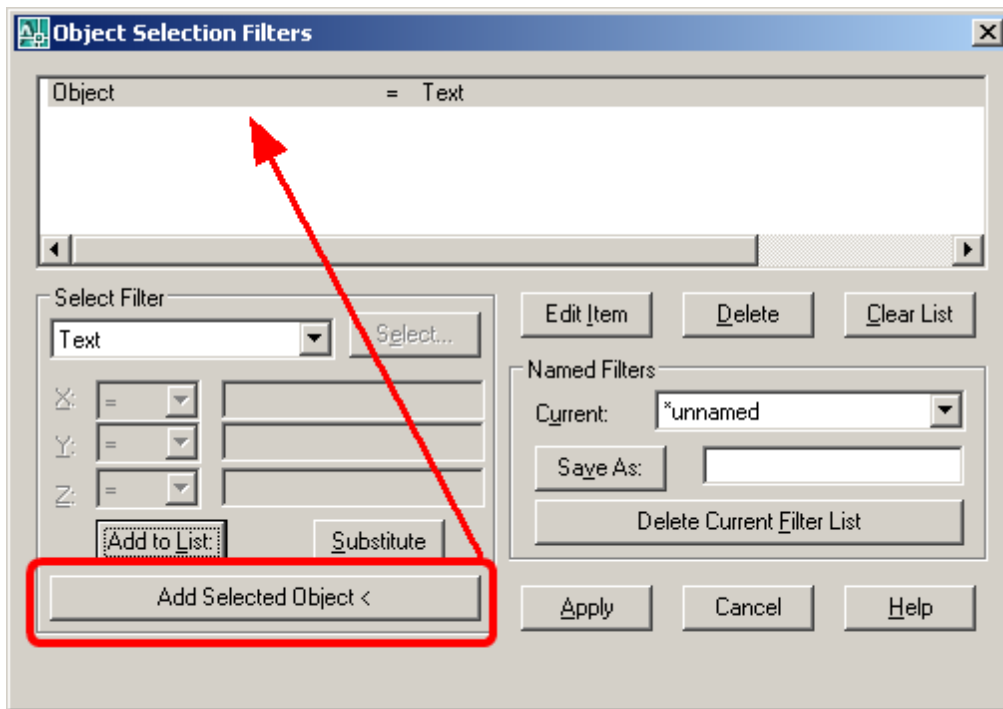
This will show the Object Selection Filters Properties window



In the Select Filter list, scroll down and pick TEXT



Then pick on Add Selected Object to add it to the list above



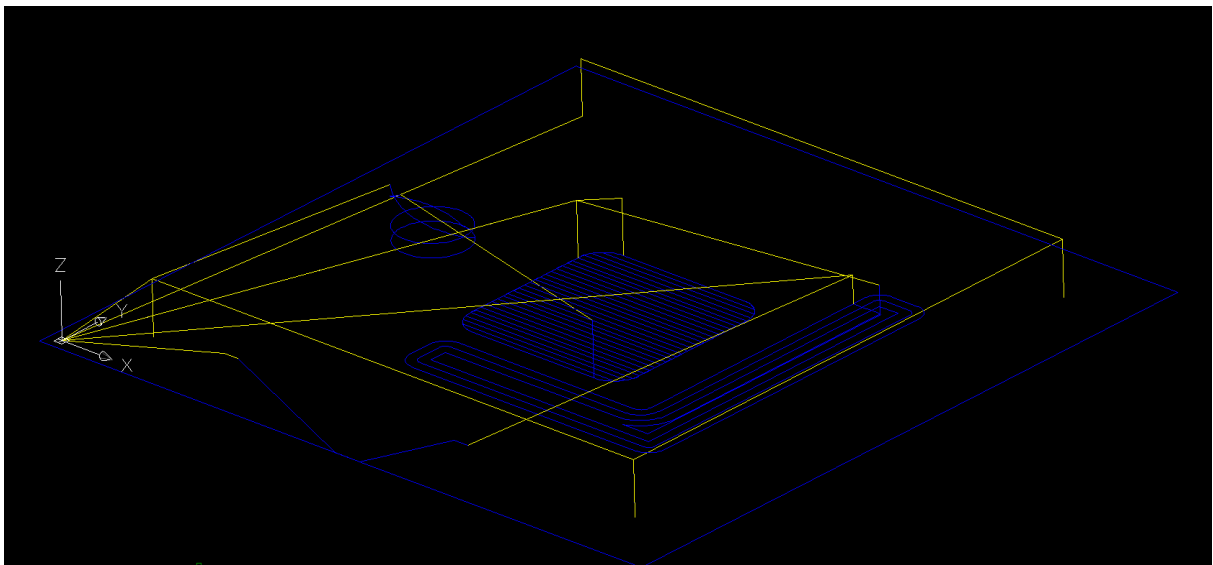
Then select Apply and you will be back at the command line.



At the Select object prompt, type in ALL

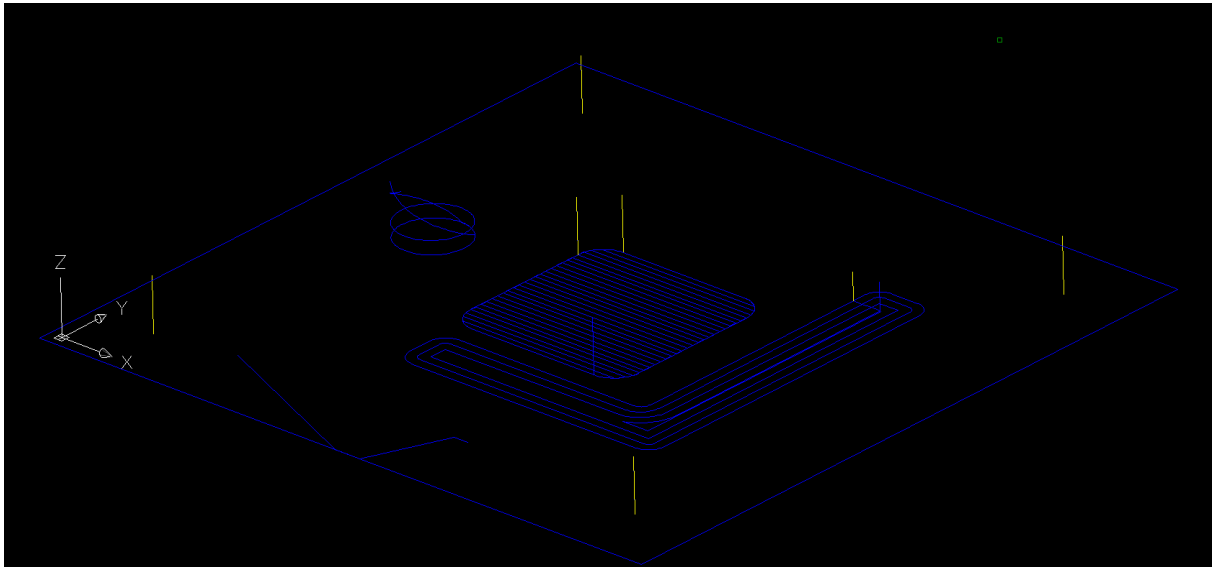


And press <ENTER>



This will remove all the text statements from the drawing and leave you with the tool moves and index lines.

If you erase the index moves, you have something closer to the part you cut.

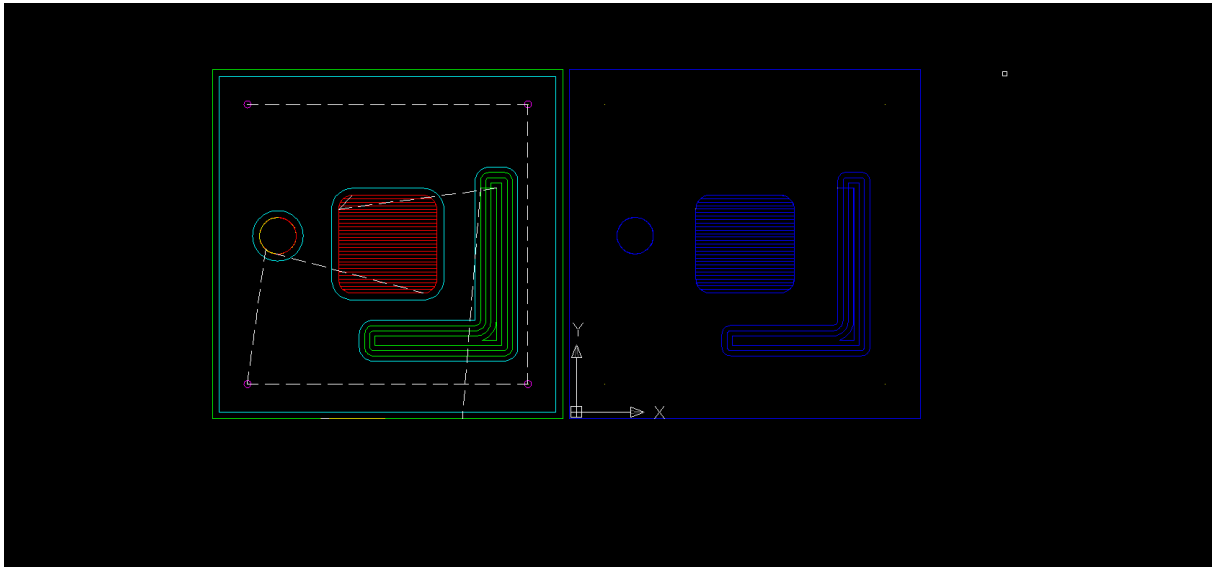


This is not exactly your part, remember that all the tool moves could be offset from your original geometry to allow for Cutter Compensation. You also do not know what size the holes in the 4 corners are, so you would have to read the code and see if it uses a common tool that you know, or has a tool comment that would show you what tool was used, like this:

```
N5M08
N6(DRILL .25 DIA.)
N7G28G91Z0M05
N8G90T2006M06
N9T102
N10M03S18000
N11G00G17G55X1.Y11.
N12G00G43H6Z1.
```

This code shows that a .25 drill bit was used, so you could draw some .25 diameter circles where the drill points are.

Looking at the backplotted geometry next to the real geometry shows the differences.



From here you can see that the outside geometry needs to be offset in and the inside cuts need to be offset outwards to recreate the actual part. You will need to do some work to make a part from code, but the general outline of what was cut will be shown.

Parametric Macro Builder Introduction

The Parametric Macro Builder (PMB) is a feature of Router-CIM to create geometry which can be inserted into AutoCAD or with the use of Router-CIM Automation Suite. The main concept or idea behind how geometry is created with this feature is to generate part drawings while locking part features into a secure position. Everything drawn begins with an initial rectangle defined by XDIM, YDIM, and ZDIM dimensions. Once the initial rectangle is established, we can create geometry constrained inside of it.

Creating a Macro consists of defining variables and formulas that can be used as parameters in the construction of specific geometric shapes. Using these defined variables, forms are provided that define the parameters for different predefined geometric shapes. The Macro is a collection of these different geometries defined as a list of operations. Also, additional logic statements can be added or inserted between geometry operations that can control variable values which can affect the parameters of geometry operations.

So, a Macro can be viewed as a collection of geometry operations and a collection of logical operations that are sequenced in a list to provide geometry development based on different logical decisions. An example sequence in a Macro would be:

- A List Variable names with formulas - Logical Operation
- Panel - Geometric Operation
- CutOut - Geometric Operation
- A new List of Variables with formulas or redefining previous variables that will affect the following operations - Logical Operation
- ToeKick - Geometric Operation
- Drill - Geometric Operation

In the above Macro sequence variables are defined in the first operation which will be used when defining the Geometric Operations. In the middle of the sequence a new set of variables are defined in another Logical Operation. This operation may redefine existing variable values and/or define new variables that can be used in the following Geometric Operations.

Who uses the Parametric Macro Builder?

A Router-CIM user that has a full understanding of programming techniques that include:

- Defining variable names that can be referenced in formulas and parameters
- Defining formulas that can include logical statements and geometric functions
- Developing sequences of operations in a logical order to utilize variable definitions and logical branches

In other words, a basic understanding of algebraic expressions and program flow. Not real high level stuff but a good basic understanding of logic, formula and formats.

What are the Parametric Macro Builder Features?

Provided is a full set of tools to aid in the development of a Macro. The general list is:

- **Operations Explorer** - keeps a complete detailed list of operations and their primary definitions. Ease of selecting, browsing and altering the sequence using the Explorer.
- **Operations Editor** - easy to read forms with predefined geometric operations. Provides simple access to variables with drag and drop along with clipboard cut, copy and paste.
- **Operations Viewer** - Full color 3D rendering of the Macro real-time. Each added operation automatically appears in the Viewer. 3D Orbit, Panning, Zooming is available with Selection.
- **Available Variables Viewer** - A pull-down window that is always available for easy access to variables that can be dragged and dropped onto forms.

- **Simple Stroke Interface** - All features are supported by easy mouse buttons/wheel motions and key board strokes.
- **AutoCAD Drawing** - Macro execution as a AutoCAD drawing generation for technical viewing and analysis.
- **3D Solids** - 3D Modeling of the Macro to view the geometric operations as they actually will be generated.
- **Macros Management** - Complete Filing of Macros and Database management of Macros to include Open/Save/Import/Export.
- **Object Orientated** - Take features and/or variables from one Macro and apply them to another Macro for fast development using previously defined information. No duplication of effort.

Many more are defined within this help document in the appropriate topics.

Macro Builder Dialog Interface

Understanding the different aspects of the dialog interface provided is important to all topics in this document. Take the time to understand and experience the different aspects of the interface before attempting to use any of the features of the Parametric Macro Builder.

Starting the Parametric Macro Builder

Router-CIM Toolbar: 


Router-CIM Ribbon:

Keyboard: **MAC**

The Parametric Macro Builder dialog window will appear. On first start you may want to center the dialog box in your display.

The Parametric Macro Builder is actually a dialog box that is displayed by Router-CIM. All AutoCAD and Router-CIM commands are suspended while in the Macro Builder.

The current Macro that appears will depend on three situations:

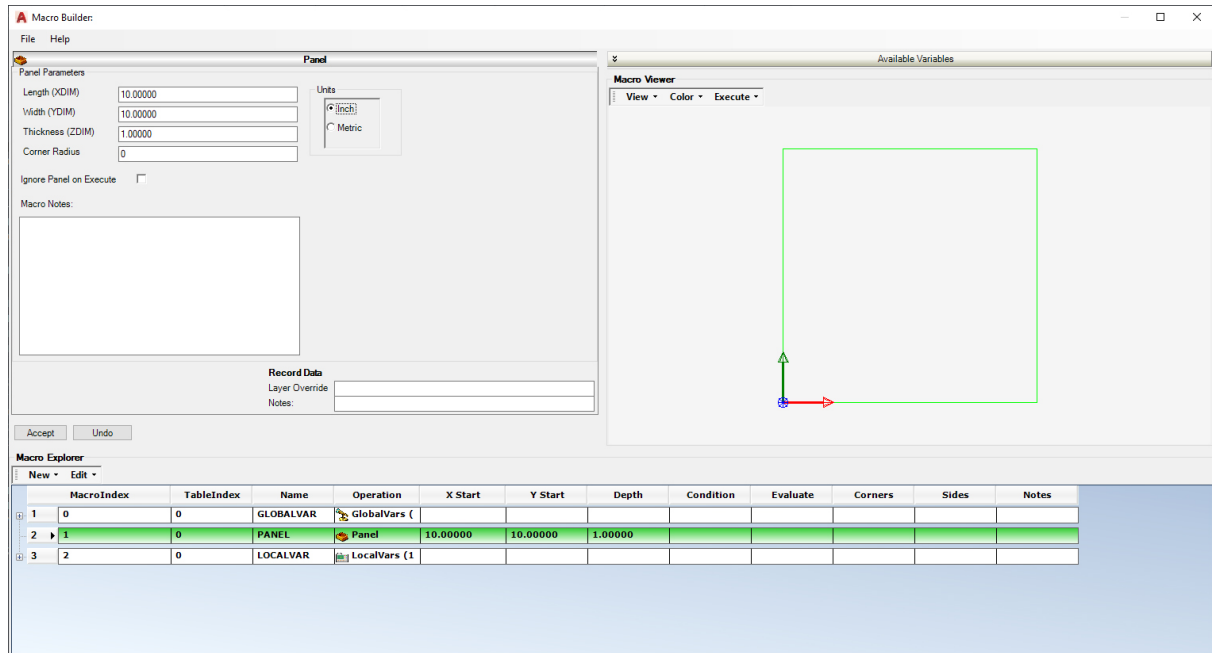
- A New Macro will be loaded if a Macro has never been referenced before by the Macro Builder.
- The last Macro that was viewed by the Macro Builder will be loaded. This Macro comes from the Macro Builder database that stores the last Macro viewed.
- If you have exited to AutoCAD and returned to the Macro Builder in the same AutoCAD session, the current Macro with any edits will appear. All views are the same as when you exited.

The Macro Builder Window

There are three distinct panes in the Macro Builder dialog window:

- **Editor** - located in the upper left
- **Viewer** - located in the upper right

- **Explorer** - located along the bottom of the dialog window.



What Macro is this?

All examples in this help references the Macro named STYLE17. To see this Macro in your Macro Builder select the File menu item, select Open, navigate to the rcim_work directory and select the **STYLE17.SCN** file and press Open in the file dialog box. The Macro will appear in the window and the Explorer will be located on the Panel Operation.

What Mouse and Keyboard features will I need to know?

Left Mouse button will select items.

Middle Mouse Wheel will scroll the Explorer items, Variable items and Zoom the Viewer.

Right Mouse Button will display pop-up menus in the Explore and Viewer and show Clipboard operations when in an edit box in the Editor.

Up/Down Arrows on the key board will scroll Explorer items and Variable items

Overview

Each pane in the dialog window works in conjunction with the other panes. When you move the mouse over each of the panes focus is given to that pane. If you select an operation in the Explorer the Viewer and the Editor will highlight that operation. If you select an operation in the Viewer then the Explorer and the Editor will highlight that operation. You can hover the mouse over a particular pane and all mouse functions defined for that pane are available. See the individual pane descriptions in the following sections for available mouse and keyboard functions.

Left Mouse Button

The left mouse button is used just like a normal select. All individual selection of any type in any pane is done with the left mouse button.


Mouse Wheel

All grid displays support the mouse wheel. For example, the Explorer is a grid and when the mouse is over the Explorer you can use the mouse wheel to scroll through the operations. Scroll bars are also available on the right side of all grids.

Two modes for the Mouse Wheel

When scrolling using the mouse wheel on any grid in the Macro Builder there are two modes available for scrolling:

- **Scroll without Select** - just like using the side scroll bar. This is the default behavior.
- **Scroll with Select** - just like using the up/down arrow keys on the keyboard

To switch modes use the mouse wheel as a button. Hold down the mouse wheel until you see the  cursor then release the wheel. It should be just like a mouse button click. When that wheel click is made then the scroll mode has been switched. To switch back to the other mode perform the same wheel click.

When in Scroll with Select mode and you rotate the mouse wheel over the Explorer each operation is selected on each click of the wheel. The Viewer and Editor are updated with the selected operation at the same time. When in Scroll without select scrolling will be the same as using the scroll bar on the side of the grid.

Right Mouse Button - Context Menus

All three panes support right click context menus. Each pane has its own menu for correct usages in that pane. See individual pane descriptions for details.

Keyboard Up/Down Arrows

When focused on any grid in the Macro Builder the Up and Down keys on the keyboard can be used to select up and down the grid. These keys are always in the Scroll with Select mode.

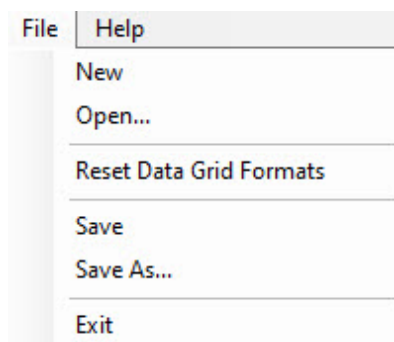
File & Database Menus

File Management

A Macro can be saved to disk as a file. The extension name given to a Macro file is **scn**. For example, the Macro Style17 would be saved to disk as STYLE17.SCN.

All file dialogs are your standard Windows Open and Save file dialogs. When you determine the directory to store a Macro that directory is added to the Macro Builder directory list and will be the directory of choice each time you use any of the File utilities.

Open, **Save** and **Save As** menu selections retrieve and store Macro files to disk.



Exit closes the Macro Builder dialog. The Macro Builder is still available in the state as you left it. You can leave the Macro Builder at anytime and return will no changes to the Macro information or edits. If you are in the middle of an edit, an Edit Pending warning is displayed and the Exit request is cancelled.

New creates a New Macro in the Macro Builder replacing its definition with the current Macro. The current Macro is discarded. If edits have been made to the current Macro a message asking "Discard Current Edits" is displayed.

Yes - discards all changes to the current Macro. A new Macro is created.

No - prevents the new macro from being made and leaves the current Macro in the Macro Builder.

A new Macro consists of available Global Variables, one Panel 10x10x1 and one default Local Variable.

Explorer

The Explorer pane displays the structure of the Macro sequence, outlines basic attributes of each operation, provides for addition of new operations, and edits the order of the Macro sequence.

In this section we cover the topics of:

- 1) [Explorer Interface](#)
- 2) [Explorer Menu Options](#)

Explorer Interface

An operation's attribute is displayed for reference only. To edit any operation's information use the Editor when the operation is selected. The Macro sequence is displayed in a grid form. Each operation is listed

in the order they appear in the sequence. Operation numbers are provided to the left of the grid to indicate the position in the Macro sequence. Each operation's basic attributes are displayed in the columns of the grid. Most operations share the same type of basic information so the columns reflect those basic attributes. If an operation's attribute is not reflected in the operation's information the content of that column will be blank.

Navigating the Explorer

Selecting any row in the grid selects a operation. The Editor and the Viewer will highlight the selected operation.

If the grid rows extend past the Explorer pane there will be a scroll bar on the right side that can scroll through the operations. Also, using the mouse wheel will scroll through the operations. See "The Interface" for more information on the mouse wheel usage.

Most operations consist of one entry in the Explorer. Global Variables, Local Variables, Mac Variables and Profile use multiple records to defined their complete operation. There are expansion indicators to the left of these operations in the Explorer. These expansion indicates are seen as a + sign. To expand the operation information to view all the records for these operations click on the + sign. To collapse the records select the - sign.

The grid columns can be resized by selecting the column title edges and dragging the edge. If a content of a cell is not totally visible you can hover the mouse over the cell and the entire contents will displayed.

Macro Sequence Structure

The sequence of operations has "fixed" operations and "editable" operations. The fixed operations always occur at the same location in the Macro and can not be moved in the sequence. All other editable operations can be relocated in the Macro sequence.

The first three operations in a Macro are fixed in the sequence. The operations and order are:

- Global Variables
- Panel
- Local Variables

These are fixed in the order described above and can not be relocated in the Macro sequence. The Global and Local Variables operations have multiple records in their definitions. Each always contain at least one record (whether used or not in the Macro logic).

Macro Explorer									
New Edit									
	Operation	X Start	Y Start	Depth	Condition	Evaluate	Corners	Sides	Notes
1	GlobalVars (8)								
2	Panel	18.00000	24.00000	0.75000					
3	LocalVars (103)								
4	Profile (5)	RAIL_LEFT	STYLE_BOTTO	0		✓ Yes	LL	T	
5	CutOut					✓ Yes		T	
6	Profile (4)	FIRSTPTX	FIRSTPTY	0		✓ Yes	LL	T	
7	Profile (4)	FIRSTPT3X	FIRSTPT3Y	0		✓ Yes	LL	T	
8	Profile (2)	FIRSTPTX-VA	FIRSTPTY	0		✓ Yes	LL	T	
9	Profile (4)	FIRSTPT2X	FIRSTPT2Y	0		✓ Yes	LL	T	
10	Profile (4)	FIRSTPT4X	FIRSTPT4Y	0		✓ Yes	LL	T	

In the above image the current selected operation is highlighted in the grid.

Macro Explorer								
New Edit								
Operation	X Start	Y Start	Depth	Condition	Evaluate	Corners	Sides	Notes
4 Profile (5)	RAIL_LEFT	STYLE_BOTTO	0		✓ Yes	LL	T	
Entity	X Start	Y Start	X End Parm.	Y End Parm.	Method			
1 Line	RAIL_LEFT	STYLE_BOTTO	RAIL_LEFT	STYLE_BOTTO	SP EP			
2 Line	2	3.75	RAIL_LEFT	STYLE_BOTTO	SP EP			
3 Line	2	5.5	RAIL_LEFT+HL	STYLE_BOTTO	SP EP			
4 Line	6	5.5	RAIL_LEFT+HL	STYLE_BOTTO	SP EP			
5 Line	6	2	RAIL_LEFT	STYLE_BOTTO	SP EP			

In the above image the current selected operation is the first record in a Profile operation using the expand grid indicator. See the Profile Operation section for the description of information.

Operations Information

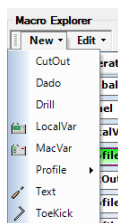
Each column heading represents an attribute of an operation. The cells in a column represents the value of that attribute. If an attribute is not associated to a particular operation then the cell value for that operation will be blank. The column headings and definitions are:

- **Operation** - the name of the operation. (xx) indicates the number of records available in the operation. Select the expansion indicator to view the records.
- **X Start** - X start location of a geometric operation. Defined as a formula, variable name or number.
- **Y Start** - Y start location of a geometric operation. Defined as a formula, variable name or number.
- **Depth** - Z depth of a geometric operation. Defined as a formula, variable name or number.
- **Condition** - a logical formula that can evaluate to True or False as a result. Determines whether the operation will be evaluated, render and executed.
- **Evaluate** - the result of the condition. If the condition evaluates to True then Yes is shown. Otherwise, No is shown.
- **Corners** - corners used as references in the definition of an operation. LL - Lower Left, UL - Upper Left, UR - Upper Right, LR - Lower Right of the Panel definition.
- **Sides** - the sides of the Panel the operation is executed on. T - Top, L - Left, B - Back, R Right, F - Front of the Panel.
- **Notes** - text notes that can be added to the operation definition in the Editor.

Explorer Menu Options

Adding New Operations to Macro

In the Explorer pane is a **New** menu that provides a list of all the available operations that can be added to a Macro sequence. Reference each operation's definition in this Help document for the individual operation description.



By selecting the **New** menu and then selecting the required operation, a default operation definition of the type selected is added to the end of the Macro sequence.

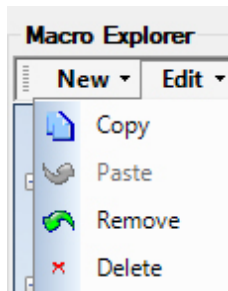
Default Profile
Selected Polyline

- **Default Profile** - generates a single line that starts a 0,0 and ends at 1,1.

- **Selected Polyline** - this option is available if a polyline has already been selected in AutoCAD. To perform the selection: 1) Exit the Macro Builder, 2) Select a polyline by Double-Clicking on the polyline, 3) Run Mac again and this option will be available.

Editing the Macro Sequence of Operations

The **Edit** menu provides for several different methods to edit the Macro sequence. These methods are available or not available depending where in the Macro sequence the edit is being made. When an **Edit** method is not correct for the operation selected, the method will be "grayed out" and not selectable.



These **Edit** methods are not to be confused with the Windows Clipboard functions. The changing of a Macro sequence is controlled by the Macro Builder using its own internal functions. Even though the methods are similar to Clipboard functions **these Edit methods are not Clipboard functions**. You can not Copy operations and paste them into another application.

These Edit methods provide for the movement of operations within a Macro sequence and the ability to Copy/Insert operations from one Macro to another Macro.

All Edit methods support multiple selection of operations. Multiple selection is performed by hold the Shift or Ctrl key while selecting multiple operations.

Editing Methods

- **Copy** - will make a copy of the selected operations and store those operations in a temporary buffer to be inserted back into the same Macro or another Macro.
- **Paste** - if the temporary buffer contains operations that have been copied or removed, this will insert the collected operations at the (before) current selected operation.
- **Remove** - will place the selected operations into a temporary buffer and remove the selected operations from the Macro. These operations can be insert back into the same Macro or another Macro.

- **Delete** - will delete the selected operations from the Macro permanently. A Yes/No question will be asked to validate that you want to delete. No Undo is available.

Move selected operations is available by using the Remove method and then using the Insert method at the desired location..

Importing particular operations of a stored Macro - Open a Macro from File or Database, Delete those operations you do not want, Save the edited Macro to File or Database using a new Macro name.

Viewer

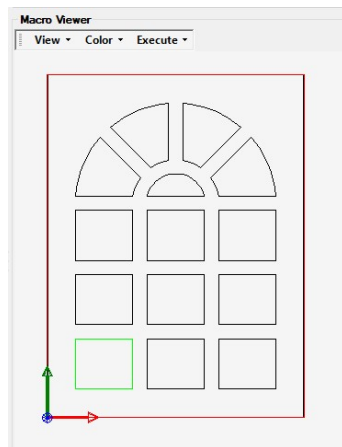
The Viewer pane displays the graphic results of the evaluated geometric operations and can also execute AutoCAD geometry and/or a 3D Solid that represents the Macro Operations.

In this section we cover the topics of:

- 1) [Viewer Interface](#)
- 2) [Execute Drawing and 3D Solid](#)

Render Interface

The Viewer displays the graphic results of the evaluated geometric operations.




Features available include:

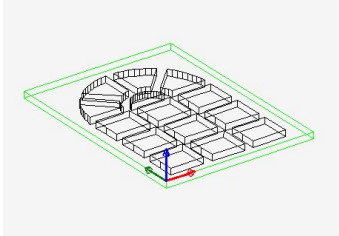
- Highlighting of currently selected operation
- Visual selection of an operation
- 3D orbit rotation in a single mouse motion to view all sides of the Macro
- Panning
- Zooming
- Full control of color options for all aspects of the operations and the display
- Right-Click shortcut mouse menu for ease of navigation
- XYZ Axis indicators

X Axis - Red axis indicator
Y Axis - Green axis indicator
Z Axis - Blue axis indicator

By placing the mouse over the Viewer activates the Viewer commands.

Selecting an Operation - position the cursor over the desired operation and click the left button. The operation will change to a highlighted color and the Explorer and Editor will select the operation. Selection can occur at any rotation, scale or panned location.

3D Orbit Rotate - position the cursor over the image and press and hold the left button. The  cursor will appear.



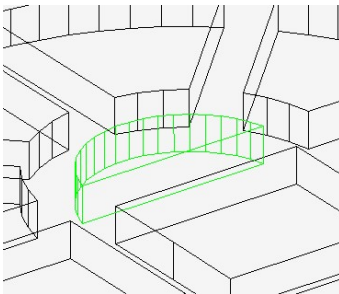
Use one of the following methods to orbit around the objects:

- To rotate along the XY plane, drag the cursor left or right
- To rotate along the Z axis, drag the cursor up and down.
- To rotate using both XY plane and Z axis, drag the cursor in a diagonal direction back and forth.


Release the left button to stop the rotation.

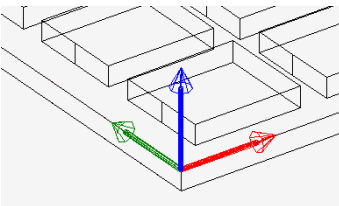
Shown view was rotated by moving the cursor from the lower left to the upper right for a short distance.

Zooming - by rotating the mouse wheel the image is zoomed in and out. Forward rotation of the wheel zooms in and backward rotation of the wheel zooms out.



Shown view was zoomed in by rotating the mouse wheel forward. Also, the left button was clicked over the half-moon feature and is selected. The selected operation is drawn in the highlighted color.

Panning - by pressing and holding down the mouse wheel and moving the mouse panning is performed. A pan moves the operations in the direction of the mouse movement. When panning the cursor changes to a  image.

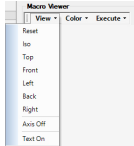


Shown view was panned down to view the axis indicators.

Right-Click Short Cut Menu - a right click while on the image will show a menu of options to aid in viewing the Macro from different pre-defined directions.

View Options include:

- Reset - resets the view to top with no zoom or pan.
- Iso - Isometric view
- Top - Top view
- Front - Front view looking along the positive Y axis
- Left - Left view looking along the positive X axis
- Back - Back view looking along a negative Y axis
- Right - Right view looking along a negative X axis
- Axis Off - turns off the axis indicators. Toggles Off/On

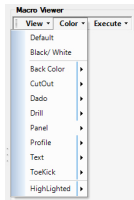


See [Text Operation](#) for "Text On" option.

Color Options - all operations and image colors are user selectable. Any settings of these colors will be saved in the Macro Builder and will be used each time the Macro Builder is run. A reset to the default settings is available.

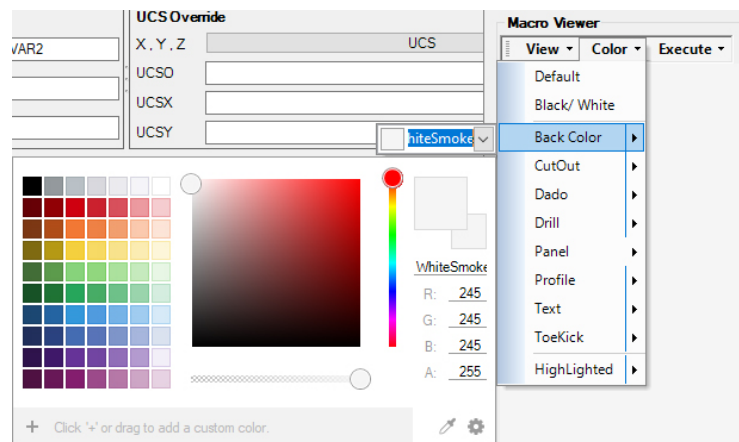
Color selection for all items in the image.

The left image shows the menu options.



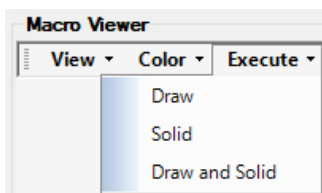
The right image shows the color selection menu.

All color changes are displayed immediately upon selection.



The color selections used for the Viewer are also the colors that will be used for drawing operations in AutoCAD. See "[Execute Drawing and 3D Solid](#)".

Execute Drawing and 3D Solid



Execute develops AutoCAD geometry and/or a 3D Solid that represents the Macro Operations. The Execute is of particular importance to Router-CIM Automation. The layer names used can be directly linked to cutting knowledge in Router-CIM Automation. See Router-CIM Automation documentation for details. The Macro Builder has a default set of layer names used to reference cutting knowledge.

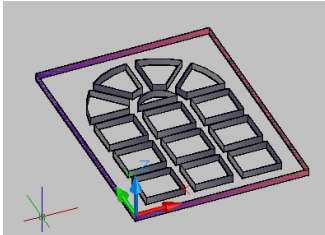
It may be necessary to define different layer names through the Editor Layer Override feature to meet your automation needs.

- **Draw** - each operation is created using AutoCAD polylines. The assigned color is given to the polylines
- **Solid** - each operation that is closed (or can be made closed) and has depth is subtracted from the Panel solid to produce a 3D Solid that represents the Macro.
- **Draw and Solid** - both the Drawing and the 3D Solid are produced in AutoCAD.

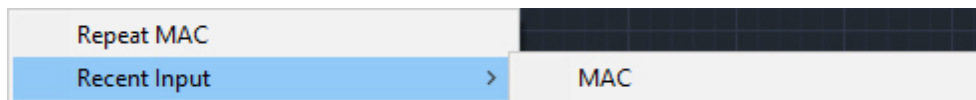
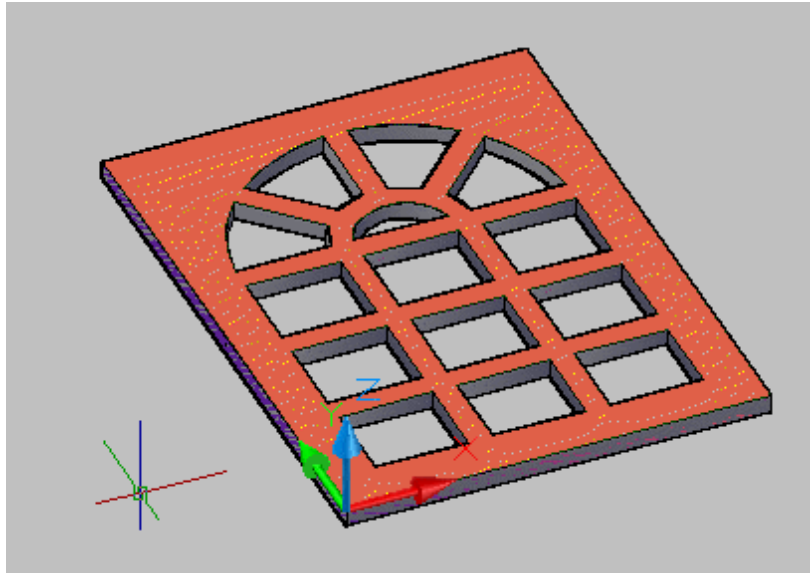
When an execute is performed the Macro Builder dialog is closed and the AutoCAD screen is available. All AutoCAD functions are available to investigate the results. If a previous Execute was performed and the drawing is in AutoCAD, those objects are deleted and replaced with the new executed objects.

The 3D Solid features use the same colors as the operation that represents the feature. If both Draw and Solid is used the polylines for Draw or "on top" of the 3D Solid for ease of feature selection.

Example AutoCAD Draw



Example AutoCAD 3D Solid



To
return
to the
Macro
Builder -
Right-
Click
in

AutoCAD and select "Recent Input" and select "MAC". If not in the list, type MAC at the command line.

Editor

These topics should be used as reference for all Editing of Macro Operations.

The Editor pane is where all the work is done. After selecting an operation or creating a new operation you will use the Editor to change the parameters for the selected operation. Each operation is unique and will have its own set of parameters. Some operations may share similar edit fields when they require the same type of parameters. Basically, the edit fields for an operation consists of:

- Text fields for values and formulas
- Graphic areas for the selection of referenced corners and sides of the Panel where the operation will be defined.
- Buttons that will perform specific tasks related to the selection operation.

In this section we cover the topics of:

- 1) [Editor Interface](#)
- 2) [Defaults and Validation](#)
- 3) [Sides, Corners and UCS](#)

Editor Interface

Reference each Macro Operation definition for details of specific edit fields.

Common Editor Fields

There are areas in the Editor pane that are common to all operations. They are:

- Accept, Undo and Reset buttons
- Condition text field and a Condition Builder button
- Layer Override field
- Notes field

Condition	Record Data
<input type="text"/> <input type="button" value="Cond..."/>	Layer Override <input type="text" value="_onepass"/>
	Notes: <input type="text"/>

Text Fields

Upon any change to a text field an edit has taken place. There is no need to acknowledge the edit with Enter or any other key. The TAB key will position to the next text field.

Accept, Undo, Reset

When any change to a parameter is made the Accept and Undo buttons will appear immediately. You should make all edits to any or all parameters before accepting the changes. Press the Accept button to accept all parameter changes. While positioned on the same operation you have the opportunity to Undo your edits. Pressing the Undo button will undo all edits made since the last Accept. After you accept the edits an evaluation of the Macro will be done and the results will appear in the Viewer. You have one addition chance to undo those edits by selecting the Reset button. If you move to another operation Reset is assumed and no reset will be available. Accept, Undo and Reset will cause a complete evaluation of the Macro and the results displayed in the Viewer.

Condition text

Operations can be conditional. This means a condition can be defined to determine whether an Operation will or will not be evaluated, rendered or executed. If a condition is defined and the results is False then the Operation will be ignored. It will not go through the process of evaluation and therefore will not be available for rendering in the Viewer or be able to be executed. If the results of the condition is True then the Operation will be evaluated, rendered and can be executed.

There is a check or X mark that appears to the right of the Condition field that indicates the state of the condition. Either it will be check True or X False. Also, the evaluate column in the Explorer will indicate this as well.

There are two types of Conditions that can be built:

- Simple Condition - a basic condition consisting of a variable name, operator, variable name. The Condition Builder will build this condition through a dialog. See below.
- Complex Condition - a Lisp expression of any type that results in a T or Nil value. A full understanding of Lisp will aid you in building this expression. See AutoCAD Developer Help, AutoLisp functions.

See the Condition Builder below for the format of a Simple Condition.

An example of a Complex Condition using a Lisp expression: (IF (> VALUE1 25) NIL (IF (= VALUE2 10) T NIL))) This example reads as IF VALUE1 > 25 THEN FALSE ELSE IF VALUE2 = 10 THEN TRUE ELSE FALSE. The expression can be as complex as needed up to 128 characters. Also, you can substitute an expression with a variable name that is a logical lisp expression with a limit of 1024 characters. See [LocalVar Operation](#) for the definition of a complex logical lisp expression.

Condition Builder

The **[Cond...]** button will display the Condition Builder:

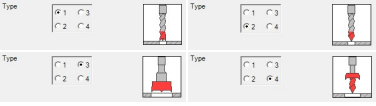
The Condition Builder consists of three fields. The "If Condition" variable, the Operator, the "Result" variable. The first variable is to be compared with the second variable using the operator. Example: A > B. A is the "If Condition" variable, > is the operator, B is the "Result" variable.

Each of the two variable fields are pull-down menus that provide you with the list of currently defined variables for selection. The Operator field is a pull-down menu of available operators. Also, if you just wanted to enter something like 1 > 0 you could type a 1 for the first variable, select the > as the operator and enter a 0 for the last variable. You can also type in the Condition text field directly the condition and skip using the Condition Builder. OK accepts the condition and Cancel ignores the edits.

Layer Override

When an operation is execute the geometry is defined on an AutoCAD named layer. The name is used for "layer to knowledge" processing in Router-CIM Automation. The layer name is crossed referenced to a particular named cutting knowledge to perform tooling instructions. There is a default layer naming convention that is used by the Macro Builder. The layer naming convention is:

Operation Type	Layer Name	
Panel	Panel_	<p>The chart shows the default layer names when the operation is defined in the Top plane (see Panel).</p> <p>When an operation is defined in any other plane the plane name is added to the default name. Example: CUTOUTRIGHT is a CutOut operation on the right side of the Panel.</p> <p>When a Layer Override name is defined that text is added to the default layer name. Example: CUTOUT_ONEPASS is a CutOut operation in the Top plane with a defined layer override text of _ONEPASS.</p>

Dado	Dado	<p>If the CutOut is on the right side of the Panel then the layer name would be CUTOUTRIGHT_ONEPASS.</p> <p>You should always check the layer name given to an operation when it is executed to ensure that you match the correct cutting knowledge to the correct layer name.</p>
Saw	Saw	
ToeKick	ToeKick	
CutOut	CutOut	
Drill	<p>Hole</p> <p>The Hole layers will end with _1, _2, _3, or _4 even with a layer override depending on the type of drill operation selected:</p> 	<p>When creating an Island type of pocketing feature, the Layer Override must begin with the word POCKET. All features for the boundary and island would need to use the same operation and same Layer Override. When the layer override begins with POCKET, it will become the beginning of the layer name with the operation being moved to the end of the layer name for use in Router-CIM Automation Suite. For example, using a PROFILE with a layer override of POCKET_MDF, the resulting layer name would be POCKET_MDFPROFILE.</p>
Profile	Profile	
Pocketing	Pocket	
Text	MacroText_	
Text Geometry	MacGeoText_	<p>This layer prefix is used when the 'Convert Text to Geometry' feature is checked.</p>

Notes

Text that may be provided to give a description to the operation.

The Editor Text Box Usages

Drag & Drop Available Variables

Most Operations have text boxes to enter values. These text boxes can be used to type in appropriate values, variables or formulas.



Note: When located in a text box, use a **Mouse Wheel Click** and the Available Variables list will pull down. This is a toggle. Click again and the list will roll-up.

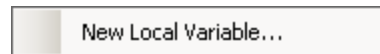
Many times the input to a text box is a predefined variable name. The text box supports Drag & Drop from the Available Variables list that can be pulled-down over the Viewer. The Available Variables list contains all the currently defined variables. By selecting a variable and dragging the cursor to a text box the variable is placed into the text box.

On-the-Fly New Local Variable Definition

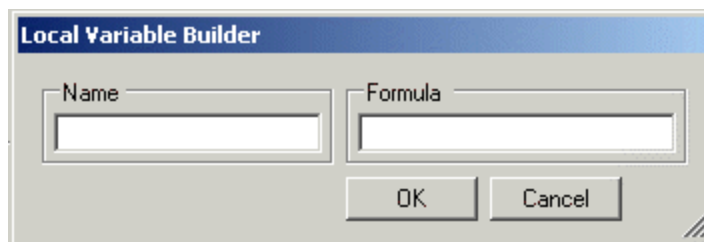
There are times when determining what values, formulas or variables to use it becomes necessary to define a new Local Variable. Instead of leaving the editing of the Operation and going to the Local Variables record you can define the variable while editing the text box, on-the-fly.

A Right Click on a text box will show the standard Windows clipboard functions. This can be used in normal clipboard operations.

A Mouse Wheel Click on a text box will show Available Variables list and the Right Click function is changed to the On-The-Fly variable definition mode.



Mouse Wheel Click then Right Click displays the Local Variable Builder.



Enter a variable Name and Formula and press OK. The new variable will automatically added to the Local Variables list and the variable name will be placed in the current text box.

To return to the standard Windows clipboard functions using Right Click - Select the text box and perform a Mouse Wheel Click.

Operations Defaults

These topics should be used as reference for all Editing of Macro Operations.

All Operations must have a minimum definition. All new Operations are defined with default parameter values. The default of an Operation defines the minimum definition. The parameters defined in a default obviously does not represent the correct definition to satisfy your requirements. Once a new Operation is added you will edit the parameters to meet your needs.

Operations Validation

All edits of parameters in an Operation will go through a serious validation process when Accept is selected. This validation process ensures that the Operation's definition meets all the requirements for proper evaluation, rendering and execution. Any error in validation will cancel the Accept, show a list of errors found and return you to the Editor. All errors have to be corrected before the Accept can be confirmed and the edited Operation evaluated.

Each [Operation](#) Help section contains a description of the valid parameter requirements.

Sides, Corners and UCS Override

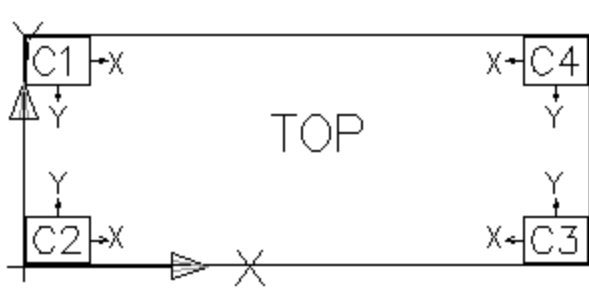
These topics should be used as reference for all Operations that provide Side, Corner or UCS parameters.

Side and Corner References

It is very important to understand the coordinate system used for geometric Operations. The selection of which coordinate system determines how the parameters are evaluated. A simple example would be a hole location at $x = +1$ and $y = +1$. The question is: what is the relationship of x and y in the defined coordinate system? If you have defined the origin of the coordinate system to be in the upper right of the Panel then $x = +1$ and $y = +1$ will be defined from the upper right of the Panel. But, is this on the top of the Panel or one of the Sides of the Panel? So, it is very important to understand the selection of a coordinate system.

Understanding Terms and Variables that define the Coordinate System

Everything references the Panel definition. The Panel defines a length in the X dimension, a width in the Y dimension and a thickness in the Z dimension. All dimensions are defined as AutoCAD World coordinates. The Top, Lower Left Corner of the Panel is 0,0,0 in World coordinates. This is the Panel origin. Note: Router-CIM automation can shift this origin when processing macros. This Help uses 0,0,0 at all times when using the Panel origin. An example Panel definition is:

Panel Parameter	Variable Name	Value	Description of Panel Variables and Reference Coordinates
Length	XDIM	18	Using the table on the left, the definition of all Panel coordinates for the TOP side can be defined: Lower Left Corner = ORIGINX, ORIGINY, ORIGINZ Lower Right Corner = ORIGINX+RIGHT, ORIGINY, ORIGINZ Upper Right Corner = ORIGINX+RIGHT, ORIGINY+TOP, ORIGINZ Upper Left Corner = ORIGINX, ORIGINY+TOP, ORIGINZ
Width	YDIM	24	
Thickness	XDIM	0.75	
Generated Parameters	Generated Variables		
Origin X	ORIGINX	0	Using corner labels C1, C2, C3, C4 the definition of the TOP side appear in the diagram below. The corner labels are not variable names.
Origin Y	ORIGINY	0	
Origin Z	ORIGINZ	0	
X - Left Edge	LEFT	0	
X - Right Edge	RIGHT	18	
Y - Top Edge	TOP	24	
Y - Bottom Edge	BOTTOM	0	

This diagram outlines the Side and Corners definitions.
The labels used are not variable names.

The corner labels are:

C1 - Upper Left (UL)

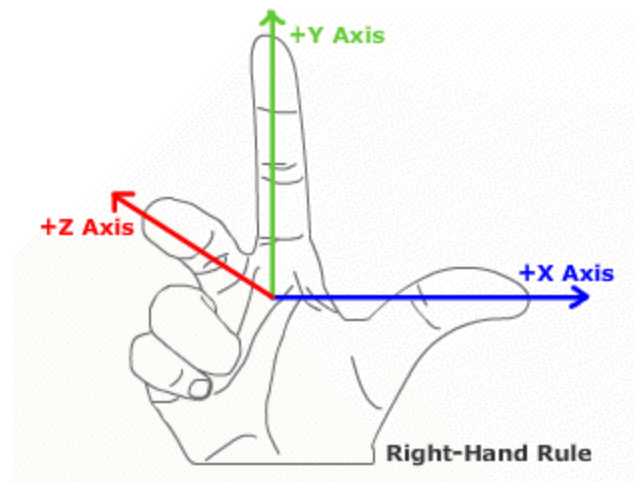
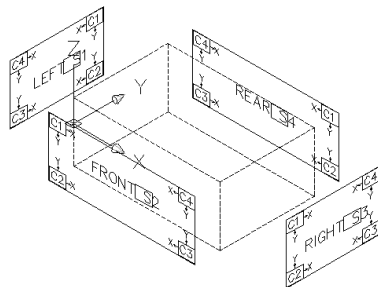
C2 - Lower Left (LL)

C3 - Lower Right (LR)

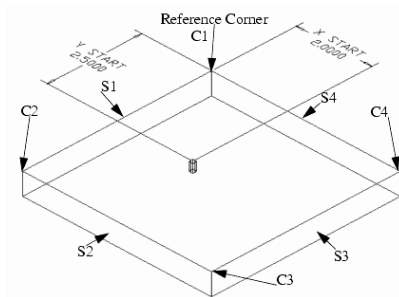
C4 - Upper Right (UR)

The X and Y axis direction is seen in the diagram.

The Z axis direction follows the right-hand rule:



Example using a Corner Reference on the Top Side



This is a simple single Drill operation.

The referenced Corner is the Upper Left.

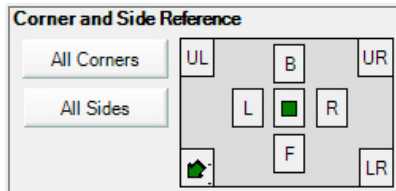
The X and Y start parameters have the values

X Start = 2.0

Y Start = 2.5

Notice that the parameters are with reference to the
Upper Left Corner of the Panel on the Top Side.

Understanding the Editor Interface for Side and Corner definitions



This example demonstrates selecting the Top Side and the Lower Left Corner as a coordinate reference.

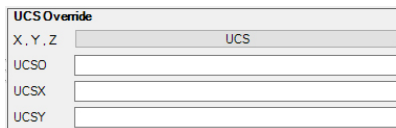
Multiple selection of Sides and Corners will duplicate the Operation on all the selected Sides and all the selected Corners

- **All Corners** - use all corners to build the operation. This results in four operations, each using a difference corner reference.
- **All Sides** - use all sides to build the operation. This results in five operations, each on a Side of the Panel
- **T, F, L, B, R** - selects the Top, Front, Left, Back and Right Side respectively.
- **LL, LR, UR, UL** - selects the Lower Left, Lower Right, Upper Right, Upper Left Corners respectively.

After selecting the desire references, press Accept and the Operation will be rendered in the Viewer.

Understanding the UCS Override

When a side is not appropriate for a plane to use for an Operation the UCS Override can be used to define the plane. A good understanding of the AutoCAD User Coordinate System (UCS) is required to use this option. Reference the AutoCAD documentation for details.



UCS button - Probably the best way to define a plane (UCS) is to use AutoCAD. Exit the Macro Builder, define a UCS and set it as the current UCS in AutoCAD to establish the plane you want to use for an Operation. Then start the Macro Builder again (MAC) and select the UCS button in the UCS Override pane. The appropriate values of the plane definition will be entered into the three available parameters.

UCSO, UCSX, UCSY - UCSO is the origin coordinate of the UCS. UCSX is the X-Direction coordinate of the UCS. UCSY is the Y-Direction coordinate of the UCS. All three parameters are required to define a UCS. The format of the coordinates is X, Y, Z values separated by commas. Variables can be used to define the X, Y, Z values.

Operations

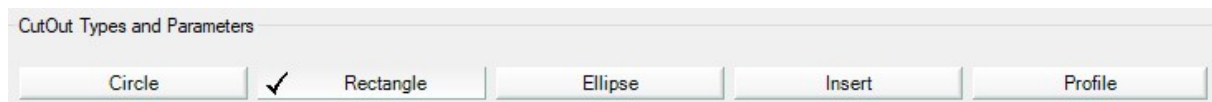
The Parametric Macro Builder (PMB) has distinct operations that it can perform. Each operation has its own unique capabilities.

In this section we cover the following operations:

- 1) [CutOut](#)
- 2) [Dado/Saw](#)
- 3) [Drill](#)
- 4) [Global Variables](#)
- 5) [Local and Mac Variables](#)
- 6) [Panel](#)
- 7) [Text](#)
- 8) [ToeKick](#)

CutOut

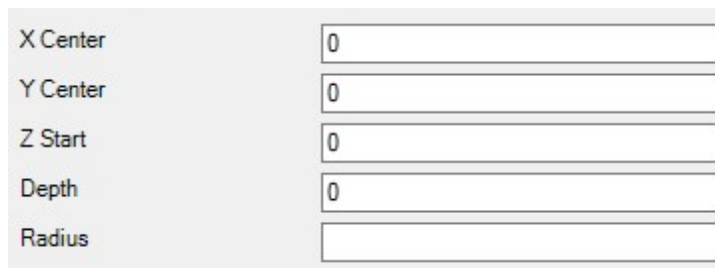
There are five different types of CutOut Operations available:



- **Circle**
- **Rectangle**
- **Ellipse**
- **Insert** - uses a defined named block in the current drawing or a named drawing file
- **Profile** - used the last defined (evaluated) Profile Operation

The checked type is the definition for the selected CutOut Operation.

Circle CutOut Parameters



- **X Center** - X location of Circle
- **Y Center** - Y location of Circle
- **Z Start** - Z location of Circle
- **Depth** - depth of Circle
- **Radius** - radius of Circle

Rectangle CutOut Parameters

X Start	<input type="text" value="0"/>
Y Start	<input type="text" value="0"/>
Z Start	<input type="text" value="0"/>
X Length	<input type="text" value="1"/>
Y Width	<input type="text" value="1"/>
Depth	<input type="text" value="0"/>
Radius	<input type="text"/>

- **X Start** - X start location of Rectangle
- **Y Start** - Y start location of Rectangle
- **X Length** - length of Rectangle in X
- **Y Width** - width of Rectangle in Y
- **Depth** - depth of Rectangle
- **Radius** - radius of corner fillets

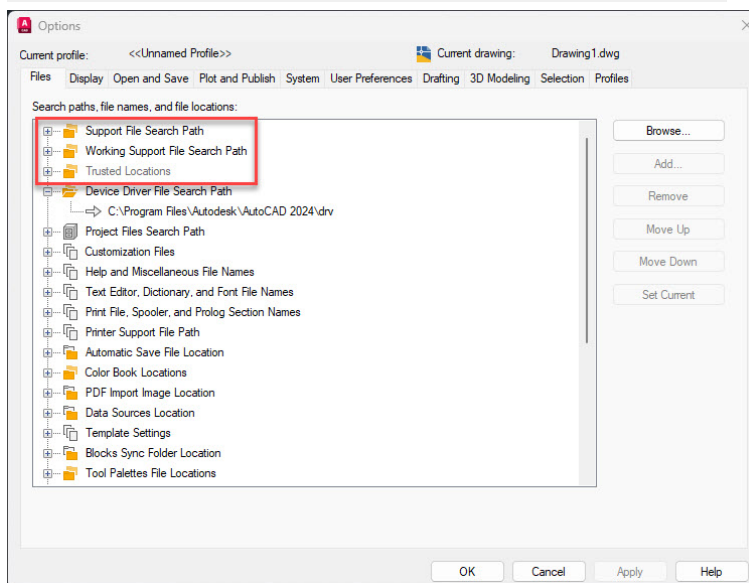
Ellipse CutOut Parameters

X Center	<input type="text" value="0"/>
Y Center	<input type="text" value="0"/>
Z Start	<input type="text" value="0"/>
X Length	<input type="text" value="1"/>
Y Width	<input type="text" value="1"/>
Depth	<input type="text" value="0"/>
Rotation	<input type="text"/>

- **X Center** - X center of Ellipse
- **Y Center** - Y center of Ellipse
- **Z Start** - Z location of Ellipse
- **X Length** - length of major axis
- **Y Width** - length of minor axis
- **Depth** - depth of Ellipse
- **Rotation** - rotation angle in degrees

Insert CutOut Parameters

X Start	<input type="text" value="0"/>
Y Start	<input type="text" value="0"/>
Z Start	<input type="text" value="0"/>
Scale	<input type="text"/>
Rotation	<input type="text"/>
Blk. Name	<input type="text"/>



- **X Start** - X location of Block insert
- **Y Start** - Y location of Block insert
- **Z Start** - Z location of Block insert
- **Scale** - scale of Block insert
- **Rotation** - rotation angle in degrees
- **Blk. Name** - name of Block
- **[Browse Blocks...]** - displays a list of block names in the current drawing for selection
- **[Browse Drawings...]** - displays a list of drawing files for selection

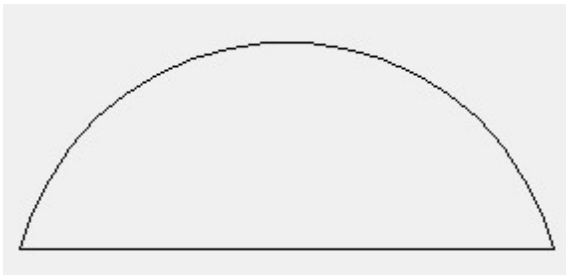
Note: Using an asterisk (*) before a variable will allow you to insert the correct DWG file as long as the DWG file is numeric (1234.dwg).

If you create a Local Variable called DRAWING_NUMBER, you would enter this in the Blk. Name field:

***DRAWING_NUMBER**

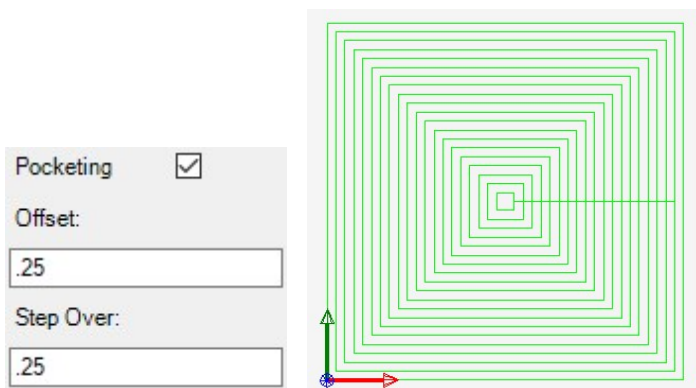
The default folder location is C:\rcim_work\HARDWARE. The folder location of the DWG file also needs to be located in the support paths in AutoCAD as shown in the image if you save the DWG files in another location.

Profile CutOut Parameters



No Parameters - a display of the last evaluated Profile in the Macro sequence prior to this CutOut Operation.

Pocketing Parameters



Any of the CutOut types can be pocketed. Pocketing is the process of developing tool motions that will remove all the material inside of the CutOut. The method of pocketing is in a Spiral Offset path.

- **Offset** - the offset distance of the tool path next to the edge of the CutOut
- **Step Over** - the distance between all other tool paths in the pocket.

Array Parameters

The array parameters are exactly the same as those detailed in the Drill Operation. Reference the [Drill Operation](#).

Validation Process

Each combination described above is validated for the correct number and types of parameters defined. The parameters must meet one of the above combinations to be accepted.

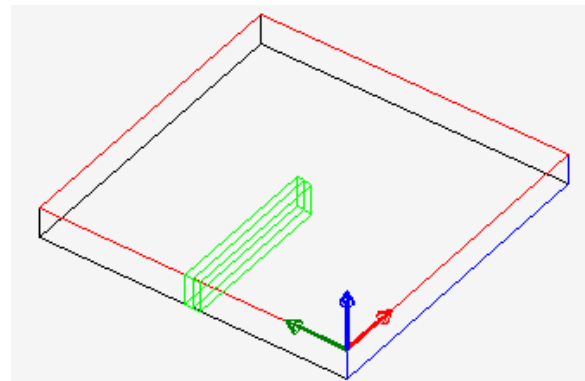
Sides, Corners, and UCS Override

Reference the [Sides, Corners and UCS](#) for details.

Condition and Record Data

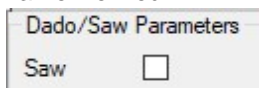
Reference the [Editor Interface](#) for details.

Dado / Saw



The Dado/Saw Operation consist of a polyline that starts at XStart, YStart and travels to XEnd, YEnd. Optional parameter definitions are:

Layer Name Defined:

Dado:  Layer name will begin with DADO

Saw:  Layer name will begin with SAW

- **XStart, YStart, XEnd, YEnd** - start at the X,Y start and end at the X,Y end parameters
- **XStart, YStart, Angle, Length** - start at the X,Y start and end at a point defined by the Angle and Length parameters

Width is the width of the Dado. Depth is the depth of the Dado. The width is visual in the AutoCAD drawing. The polyline would need a layer override in order to make sure the layer name describes the

width of the feature in order to assign the correct cutting knowledge in Router-CIM Automation Suite with the DOIT file.

Validation Process

Width and Depth must have values.

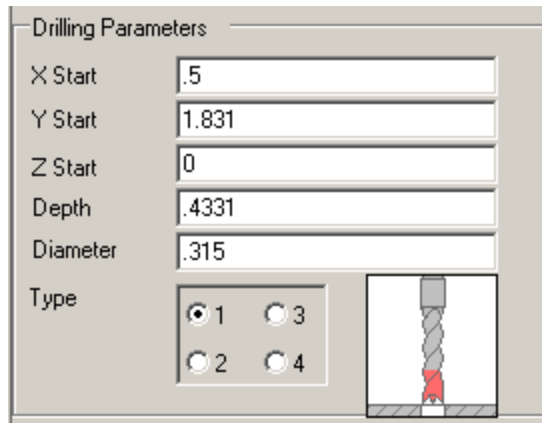
X,Y Start must have values.

X,Y End **OR** Angle, Length must have values.

Condition and Record Data

Reference the [Editor Interface](#) for details.

Drill



The Drill Operation consist a drill location specified by:

- **XStart** - X location of drill hole
- **YStart** - Y location of drill hole
- **ZStart** - Z location of drill hole
- **Depth** - Z depth of hole
- **Diameter** - diameter size of the hole
- **Type** - not used

The Drilling Parameter fields defines single Drill Operation. Multiple Drill Operations can be accomplished using the Array Parameters.

Rectangular Array Parameters

Array Type

☒ Rectangular

☐ Circular

Num Of Holes

1

X Stop

Y Stop

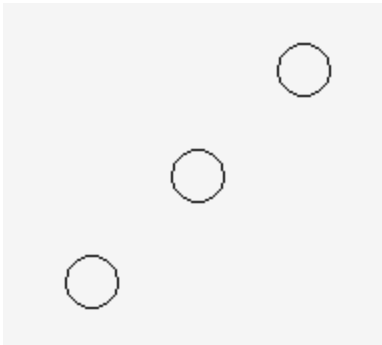
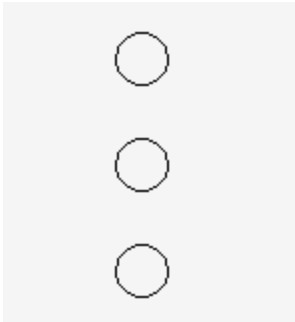
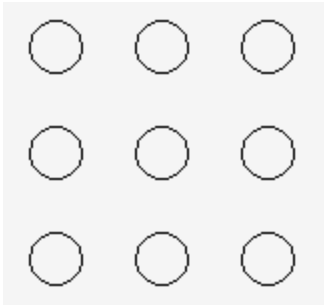
X Spacing

Y Spacing

Rows

Columns

There are four array patterns available:



- Multiple Rows and Columns array
- One Row horizontal array
- One Column Vertical array
- Single diagonal line array

Multiple Rows and Columns Array Parameters

Num Of Holes

9

X Stop

Y Stop

X Spacing

2

Y Spacing

2

Rows

3

Columns

3

OR

Num Of Holes

X Stop

7

Y Stop

7

X Spacing

2

Y Spacing

2

Rows

Columns

=

XStart and YStart = 3, Num of Holes = 9, X and Y Spacing = 2, Rows and Columns = 3 defines a 9 hole rectangular array starting at 3,3 with 3 rows and 3 columns spaced 2 units between centers in X and Y.

XStart and YStart = 3, X Stop and Y Stop = 7, X Spacing and Y Spacing = 2 defines a 9 hole rectangular array starting at 3,3 and stopping at 7,7 at 2 units between centers in X and Y.

Note: All parameters not being used to define an array must be blank.

Single Row Array Parameters

Num Of Holes	3
X Stop	
Y Stop	
X Spacing	2
Y Spacing	
Rows	
Columns	

OR

Num Of Holes	
X Stop	7
Y Stop	
X Spacing	2
Y Spacing	
Rows	
Columns	

=



XStart and YStart = 3, Num of Holes = 3, X Spacing = 2 defines a one row array starting at 3,3 and stopping at 7,3 with a 2 unit spacing between centers.

XStart and YStart = 3, X Stop = 7 and X Spacing = 2 defines a one row array starting at 3,3 and stopping at 7,3 with a 2 unit spacing between centers.

Note: All parameters not being used to define an array must be blank.

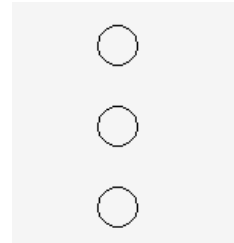
Single Column Array Parameters

Num Of Holes	3
X Stop	
Y Stop	
X Spacing	
Y Spacing	3
Rows	
Columns	

OR

Num Of Holes	
X Stop	
Y Stop	7
X Spacing	
Y Spacing	3
Rows	
Columns	

=



XStart and YStart = 3, Num of Holes = 3, Y Spacing = 2 defines a one row array starting at 3,3 and stopping at 3,7 with a 2 unit spacing between centers.

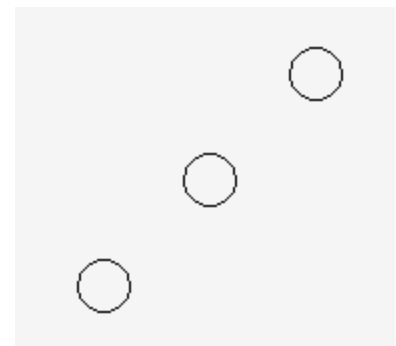
XStart and YStart = 3, Y Stop = 7 and y Spacing = 2 defines a one row array starting at 3,3 and stopping at 3,7 with a 2 unit spacing between centers.

Note: All parameters not being used to define an array must be blank.

Single Diagonal Line Array

Num Of Holes	3
X Stop	
Y Stop	
X Spacing	2
Y Spacing	2
Rows	
Columns	

=



XStart and YStart = 3, Num of Holes = 3, X and Y Spacing = 2 defines a single diagonal line array starting at 3,3 and stopping at 7,7 with a 2 unit spacing between centers.

Note: All parameters not being used to define an array must be blank.

Circular Array Parameters

Array Type

☐ Rectangular

☒ Circular

Num Of Holes

X Center

Y Center

Fill Angle

Rotate Objs ☐ Yes ☒ No

All parameters must be specified for a Circular Array:

- **Num of Holes** - number of holes in the array
- **X Center** - X center of the circular array
- **Y Center** - Y center of the circular array
- **Fill Angle** - total angle sweep in degrees
- **Rotate Objs** - rotates each hole at the angle it is at in the array.

Num Of Holes

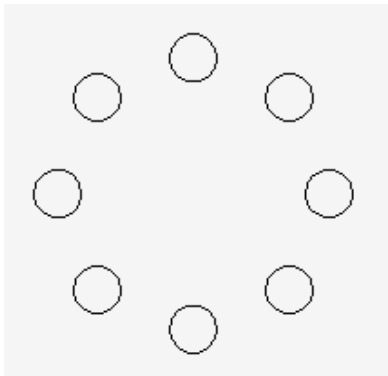
X Center

Y Center

Fill Angle

Rotate Objs ☐ Yes ☒ No

=



X Start and Y Start = 3, X Center and Y Center = 5, Fill Angle = 360, Num of Holes = 8 defines a circular array that starts at 3,3 with a center at 5,5 and with 8 holes that span a total of 360 degrees. Each hole is $360 / 8 = 45$ degrees apart from each hole center. Each hole is not rotated in the array.

Validation Process

Each combination described above is validated for the correct number and types of parameters defined. The parameters must meet one of the above combinations to be accepted.

Sides, Corners, and UCS Override

Reference the [Sides, Corners and UCS](#) for details.

Condition and Record Data

Reference the [Editor Interface](#) for details.

Global Variables Defined

The GlobalVar Operation contains all the Global variable names, associated formulas and values. Global variables are stored in a file on disk and is reference by all Macros opened by the Macro Builder.

The GlobalVar Operation is the First Operation evaluated by the Macro Builder.

Since Global Variables are the first to be evaluated there are restrictions on the formula used to define a GlobalVar.

Global Variable Formula Restrictions

- A Global Variable's formula can only reference previously defined Global Variable names or XDIM, YDIM and ZDIM Panel variables.
- A Global Variable name can not be XDIM, YDIM or ZDIM.

Local Variables are evaluated after the Global Variables. Therefore, a Local Variable is not defined while the Global Variables are being evaluated. The Panel variables XDIM, YDIM and ZDIM are evaluated prior to the Global Variables and therefore can be referenced when defining a formula for a Global Variable.

Reference the Local Variables Help section for details on Formula formatting.

Global Variables affect all Macros opened by the Macro Builder

- Global Variables are used by all Macros opened by the Macro Builder.
- Any edits to the Global Variables are always saved to disk and will appear in any Macro opened by the Macro Builder.

Global Variable Naming Conventions

- Variable names can not be a LISP function name, a CAL function name or any Protected Variable name defined by the Macro Builder
- Characters used to define a variable name must begin with a letter and can not contain special character like <blank>=</>[]{}()*+~^

Reference the AutoCAD Programmers Reference manual for Lisp function description.

Reference the CAL command in the AutoCAD User Help for CAL function names.

Reference the Router-CIM file pv.dat for a list of protected variable names. Add names to this file using a ascii editor when necessary. The validation process in the Macro Builder will use this file when validating edits on Accept.

GlobalVar Interface

GlobalVariables				
	Name	Formula	Result	Notes
1	GLB_TEST	100	100	
2	SMALLAREA	190	190	
3	FANLOC	10	10	
4	TESTVAR	20	20	
5	UV2	2	2	
6	UV3	2	2	
7	UV4	2	2	
8	UV5	2	2	
*				

Global Variables are presented in a grid format. The currently selected variable is indicated by an arrow in the numbered selector, contents are bold type and the grid row appears in a highlighted color.

To Edit an existing variable simply select the Name or Formula field and make the desired

edit. Selecting the Result field will not cause an edit. The Result field is the results determined by evaluating the Formula. This occurs after the acceptance of the edits.

To Add a New variable select the Name field next to the last un-numbered row that has an * as its selector. Enter the name, press TAB key and enter the formula.

The Name and Formula fields support the Windows Clipboard. Right Click on the field and the Clipboard menu will display.

Note: the Notes field are not saved in the global variables file. They are saved in the database.

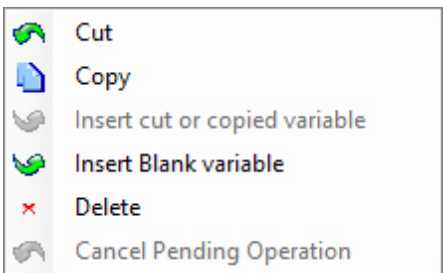
Grid Row Editing

Right Click on the Grid with display the Edit Menu.

The **Edit** menu provides for several different methods to edit the row sequence. These methods are available or not available depending prior edits. When an **Edit** method is not correct for the selected row, the method will be "grayed out" and not selectable.

These **Edit** methods are not to be confused with the Windows Clipboard functions. The changing of the grid row is controlled by the Macro Builder using its own internal functions. Even though the methods are similar to Clipboard functions **these Edit methods are not Clipboard functions**. You can not Copy rows and paste them into another application.

All Edit methods support multiple selection of rows. Multiple selection is



performed by hold the Shift or Ctrl key while selecting multiple operations.

Grid Row Editing Methods

- **Cut** - will place the selected rows into a temporary buffer and remove the selected rows from the grid. These rows can be insert back into the grid at another row location.
- **Copy** - will make a copy of the selected rows and store those rows in a temporary buffer to be inserted back into the grid at another row location.
- **Insert cut or copied variable** - if the temporary buffer contains operations that have been copied or removed, this will insert the collected rows at the (before) current selected row.
- **Insert Blank variable** - Inserts a Blank variable above the selected variable
- **Delete** - will delete the selected rows from the grid permanently. A Yes/No question will be asked to validate that you want to delete. No Undo is available.
- **Cancel Pending Operation** - if a variable is currently in 'Cut' or 'Copy', it will release it from the Parametric Macro Builder memory

Grid Cell Editing

After changing a cell content press the TAB key to accept the edit and go to the next cell in the row or press the ENTER key to accept the edit and go to the next row.

Local and Mac Variables Defined

A MacVar Operation is the same as a LocalVar Operation. However, a Macro can have multiple MacVar definitions located anywhere in the Macro sequence. The LocalVar always appears just after the Panel Operation in a fixed position in the Macro sequence. All definitions below apply to Local Variables and Mac Variables.

A Variable is defined by a Name and a Formula. The Result is determined by the evaluation of the Formula.

Parameter Definitions:

- **Name** - Ascii name of the Variable. The name of the variable cannot have spaces or dashes. Some variable names are protected and cannot be used. The Parametric Macro Builder will warn if the variable name chosen is protected.
- **Formula** - an expression that defines the Variable's result
- **Result** - the evaluated result of the Formula
- **Condition** - a logical formula that evaluates to True or False. True and False or represented as 1 or 0 in the grid. If True, the Variable will be evaluated. If False, the Variable will be skipped, not evaluated and not available for reference by any Macro Operations.
- **Status** - used only by the LocalVar Operation. Options or Tagged and Dynamic. These options are used by Router-CIM Automation suite.
- **Notes** - text description of the Variable.

Variables are evaluated in the order of position in the Macro sequence. All defined variables are available to all following Operations.

Variable Naming Conventions

- Variable names can not be a LISP function name, a CAL function name or any Protected Variable name defined by the Macro Builder
- Characters used to define a variable name must begin with a letter and can not contain special character like <blank>=/<>[]{}()+-~^

Variable Formula Restriction

- A Variable's formula can only reference previously defined Variable names or XDIM, YDIM and ZDIM Panel variables.
- A Variable's formula can be a CAL math expression, a Lisp Logical expression or a complex Lisp expression with a numeric result.

You can use the same variable name as long as a condition is used in order to only have one of the named instances available at one time.

Here is an example of using the same variable name with a condition statement.

Macro Builder:

File Help

Local Variables				
	Name	Formula	Result	Condition
1 ▶	STILE	1.75	1.75	YDIM>6
2	STILE	1.25	1.25	YDIM<=6
*				

Reference the AutoCAD Programmers Reference manual for Lisp function description.
Reference the CAL command in the AutoCAD User Help for CAL function names and expression formats.

Variable Interface

	Name	Formula	Result	Condition	Status
1 ▶	RAIL_RIGHT	2	2		TAGGED
2	RAIL_LEFT	2	2		TAGGED
3	STICK	1	1		TAGGED
4	STYLE_BOTTOM	2	2		TAGGED
5	STYLE_TOP	2	2		TAGGED
6	HLENGTH	(XDIM-RAIL_LE...	4		
7	TOPRAIL	STYLE_TOP	2		
8	RIGHTRAIL	RAIL_RIGHT	2		
9	LEFTRAIL	RAIL_LEFT	2		
10	VAR1	XDIM-LEFTRAIL...	14		
11	VAR2	STICK	1		
12	HALFVAR2	VAR2/2.0	0.5		
13	STRANGLE	0	0		
14	ENDANGLE	45	45		
*					

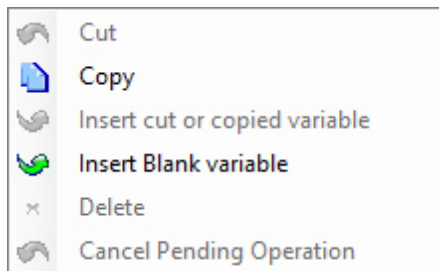
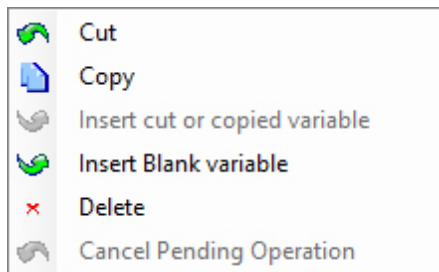
Variables are presented in a grid format. The currently selected variable is indicated by an arrow in the numbered selector, contents are bold type and the grid row appears in a highlighted color.

To Edit an existing variable simply select the Name or Formula field and make the desired edit. Selecting the Result field will not cause an edit. The Result field is the results determined by evaluating the Formula. This occurs after the acceptance of the edits.

To Add a New variable select the Name field next to the last un-numbered row that has an * as its selector. Enter the name, press TAB key and enter the formula.

The Name and Formula fields support the Windows Clipboard. Right Click on the field and the Clipboard menu will display.

Grid Row Editing



Right Click on the Grid with display the Edit Menu.

The **Edit** menu provides for several different methods to edit the row sequence. These methods are available or not available depending prior edits. When an **Edit** method is not correct for the selected row, the method will be "grayed out" and not selectable.

These **Edit** methods are not to be confused with the Windows Clipboard functions. The changing of the grid row is controlled by the Macro Builder using its own internal functions. Even though the methods are similar to Clipboard functions **these Edit methods are not Clipboard functions**. You can not Copy rows and paste them into another application. All Edit methods support multiple selection of rows. Multiple selection is performed by hold the Shift or Ctrl key while selecting multiple operations.

NOTE: If a variable name is used in a formula this dependence restricts the removing or deleting the variable name. Options to Remove or Delete the variable will be disabled as shown on the left.

Grid Row Editing Methods

- **Cut** - will place the selected rows into a temporary buffer and remove the selected rows from the grid. These rows can be insert back into the grid at another row location.
- **Copy** - will make a copy of the selected rows and store those rows in a temporary buffer to be inserted back into the grid at another row location.
- **Insert cut or copied variable** - if the temporary buffer contains operations that have been copied or removed, this will insert the collected rows at the (before) current selected row.
- **Insert Blank variable** - Inserts a Blank variable above the selected variable
- **Delete** - will delete the selected rows from the grid permanently. A Yes/No question will be asked to validate that you want to delete. No Undo is available.
- **Cancel Pending Operation** - if a variable is currently in 'Cut' or 'Copy', it will release it from the Parametric Macro Builder memory

Grid Cell Editing

After changing a cell content press the TAB key to accept the edit and go to the next cell in the row or press the ENTER key to accept the edit and go to the next row.

Formula Formats

There are three formula formats available:

- **Numeric** - a CAL formatted math expression that evaluates to a numeric result.
- **Logical** - a LISP formatted logical expression that evaluates to True or False
- **Lisp** - a complex LISP expression that evaluates to a numeric result.

Numeric

CAL evaluates expressions according to standard mathematical rules of precedence:

- Expressions in parentheses first, starting with the innermost set
- Operators in standard order: exponents first, multiplication and division second, and addition and subtraction last
- Operators of equal precedence from left to right

Numeric Expressions

Numeric expressions are real integer numbers and functions combined with the operators in the following table.

Numeric Operators

Operator	Operation
()	Groups expressions
^	Indicates exponentiation
*, /	Multiplies, divides
+, -	Adds, subtracts
SIN, COS, TAN, ASIN, ACOS, ATAN	Common Geometric Functions

The following are examples of numeric expressions:

3

3 + 0.6

(5.8^2) + PI

Reference the CAL Help in AutoCAD for a complete list of functions available.

Besides common numeric operators, you can reference the ['Common Variable Equation Expressions'](#) below for ways to compare expressions.

IMPORTANT NOTE: A CAL expression's structure is Operand followed by Operator followed by Operand; standard mathematical expressions. A CAL expression DOES NOT START WITH A = SIGN IN ITS DEFINITION. Examples: A + B or (A + B). **NO EQUAL SIGN.**

Logical

A Logical statement can be formatted in two forms:

- = A > B - formatted starting with a = sign followed by an Operand, then the Operator, ending with a Operand
- =(> A B) - formatted as a Lisp expression. Starts with a = sign to indicate a Lisp expression. See AutoCAD Developer Help, AutoLisp functions.

A logical must start with a = sign. After the = sign either format can be used.

A Logical variable formula can be formatted as a Lisp expression. The structure starts with a left parenthesis, followed by an logical operator, followed by two operands to be compared, then closing with a right parenthesis. Each separated by a space. Example: =(> 1 0) evaluated to True. 1 is greater than 0. Lisp expressions are in the form of prefix notation (postfix notation) that places the operators to the left of their operands.

An example of a Logical variable formula using a Lisp expression: =(IF (> VALUE1 25) T (IF (= VALUE2 10) T NIL)) This example reads as IF VALUE1 > 25 THEN TRUE ELSE IF VALUE2 = 10 THEN TRUE ELSE FALSE. This expression will evaluate as True or False depending on the conditions in the expression. The expression can be as complex as needed up to 1024 characters.

IMPORTANT NOTE: A LISP expression MUST START WITH A = (equal sign). This is required to indicate to the formula field that this is Lisp.

Lisp

A complex variable formula is formatted as a Lisp expression. Any lisp expression with the use of lisp functions can be used. See AutoCAD Developer Help, AutoLisp functions.

An example of a complex variable formula using a Lisp expression: =(IF (> VALUE1 25) 25 (IF (= VALUE2 10) 5 75)) This example reads as IF VALUE1 > 25 THEN 25 ELSE IF VALUE2 = 10 THEN 5 ELSE 75. This expression will evaluate as 25, 5 or 75 depending on the conditions in the expression. The expression can be as complex as needed up to 1024 characters.

IMPORTANT NOTE: A LISP expression MUST START WITH A = (equal sign). This is required to indicate to the formula field that this is Lisp.
Character limit for a LISP expression is 1024 characters.

The last evaluated expression must result in a numeric result.

A Special Variable named LISP

If you name a variable with the special name of LISP a lisp expression in the formula field is required. This Lisp expression should not start with a = sign and must start with a right parenthesis (. Any legal Lisp expression can be used and does not have to have the last evaluation be a numeric result. The Result in the Macro builder will always be zero. The LISP variable name can not be reference by any other formula in the Macro. It is a dummy name and should not be considered a Variable name that can be reference by other formulas. Example: (dofunc 5.0). This is a Lisp expression that calls a predefined function named dofunc and passes to the function one parameter 5.0.

Condition Format

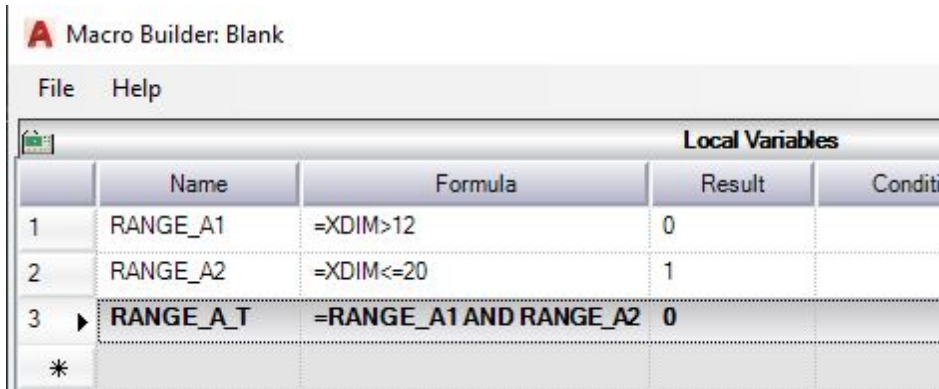
The condition format is the same as the Conditional Builder. See the Condition Builder in the [Editor Interface](#) section of this Help. An example of a condition is A > B. Notice it starts with an Operand followed by a logical Operator and ending with an Operand. No parentheses are used.

Multiple Condition Format

In order to evaluate a multiple condition format, multiple local variables MAY need to be created. This multiple condition format is common when defining a range set.

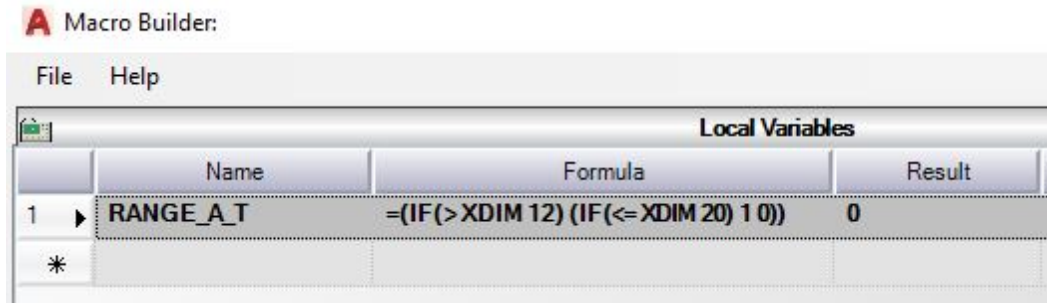
For example, if you want to have an operation that needs to be used when the XDIM is greater then 12 but less then or equal to an XDIM of 20, this would be a simple set up in the Local Variables section:

The XDIM is currently set to 10. You can see that the variable RANGE_A1 is 0 (False) but RANGE_A2 is 1 (True). The multiple condition is being checked by the variable RANGE_A_T which is 0 (False).

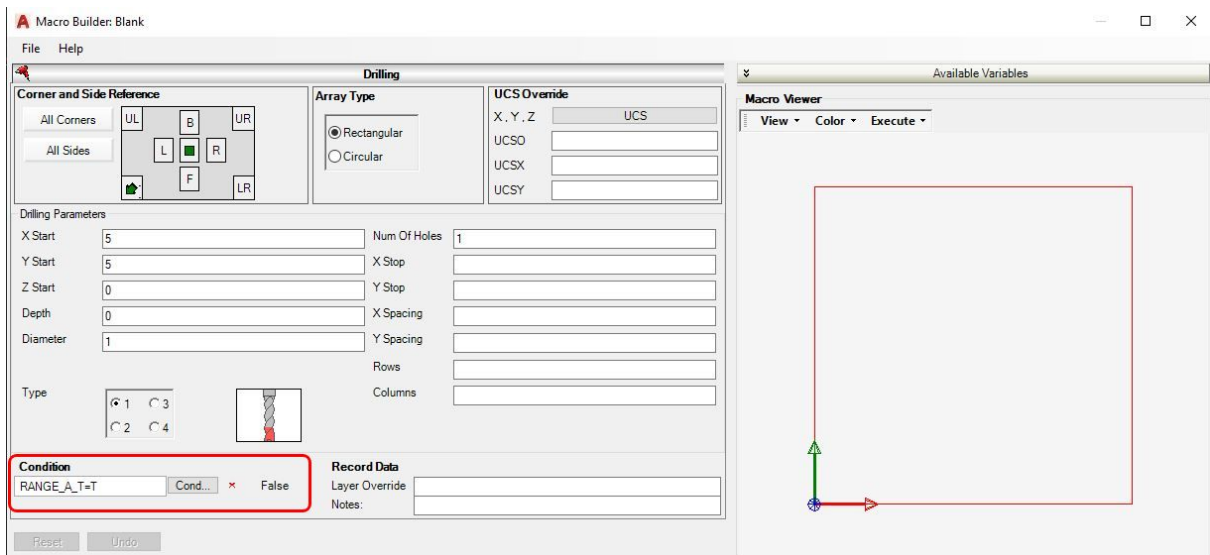


Macro Builder: Blank				
File		Help		
Local Variables				
	Name	Formula	Result	Condition
1	RANGE_A1	=XDIM>12	0	
2	RANGE_A2	=XDIM<=20	1	
3	RANGE_A_T	=RANGE_A1 AND RANGE_A2	0	
*				

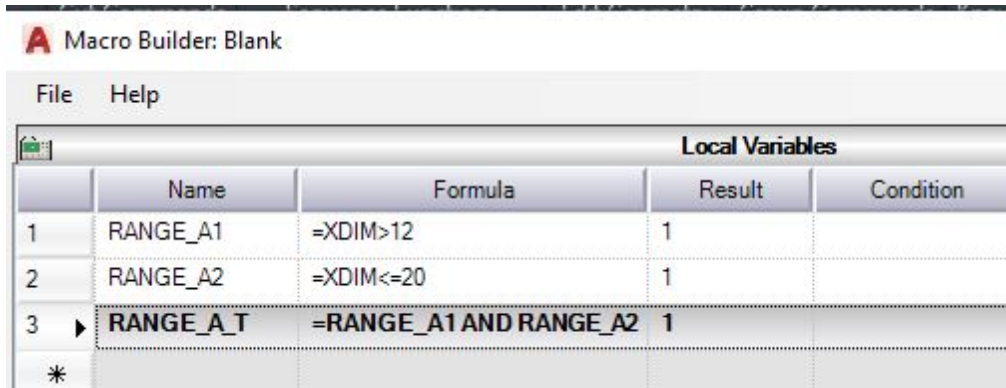
Another option for a single line formula would be =(IF(> XDIM 12) (IF(<= XDIM 20) 1 0)). This result would be NUMERIC as a 1 or 0.



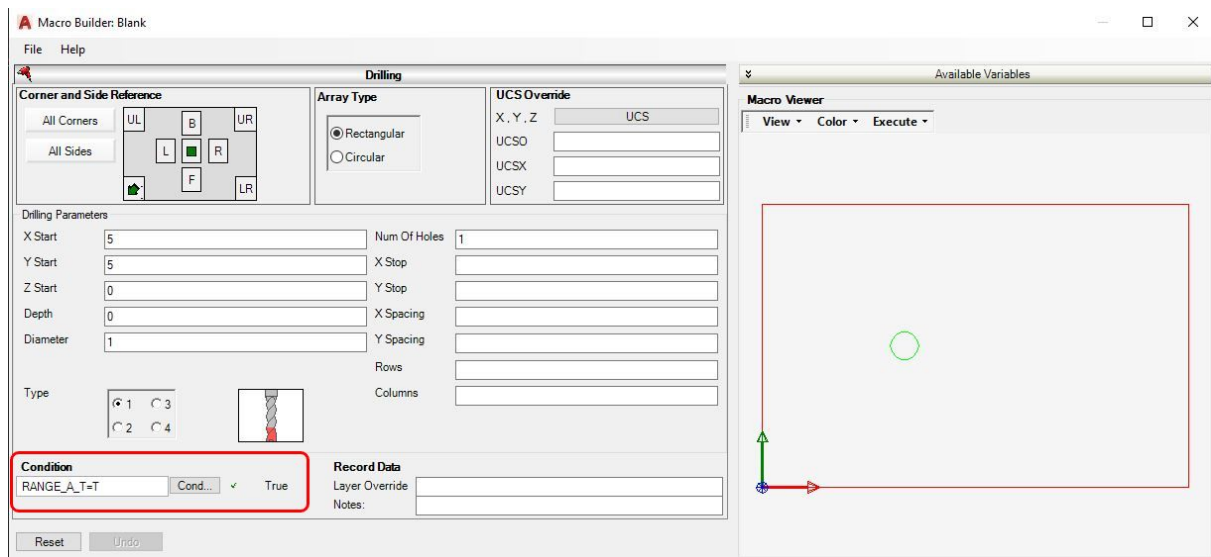
Under the Operation, you will then create a Condition statement to react to the variable RANGE_A_T as shown in the image. In this case with an XDIM of 10, the condition evaluated as False.



When the XDIM is set to 15. You can see that the variable RANGE_A1 is now 1 (True) and RANGE_A2 is 1 (True). The multiple condition is being checked by the variable RANGE_A_T which now evaluates as 1 (True).



In this case with an XDIM of 15, the condition evaluated as True turning on the operation.



Status Options

Status options include blank (the default), TAGGED and DYNAMIC. These status options TAGGED and DYNAMIC are used by Router-CIM Automation Suite.

Reference the Router-CIM Automation Suite documentation for a complete understanding of using variables in automation jobs.

TAGGED

Tagged variables are variables that can be carried to the Router-CIM Automation program with a macro. By using Tagged Variables, you would not have to go back to the Parametric Macro Builder to change the variables value.

When would I use the Tagged Variables feature and what type of variables would go into it?

For example, let's say we designed a basic door in the Macro Builder, but we want the ability to quickly change the dimensions of the door on a regular basis. By assigning variable names and proper formulas to the individual features of the door, we can quickly change the size of the door features in Router-CIM Automation Suite without going back into the Macro Builder.

Selecting the TAGGED option of Variable status in the local variables grid will cause both the name and the value of the chosen variable to be displayed in the Router-CIM Automation Suite interface when preparing automation jobs. Only these Tagged Local variables are available during Router-CIM Automation Suite.

Example:

The Variables TOEKICK_X, TOEKICK_Y, SHELVES and NUMBER_HOLES are given the values of 5, 4, 3, and 2, respectively and they are also designated as tagged. The macro has accomplished the following:

- These "tagged" variables will appear in the Router-CIM Automation Job window when this particular macro is opened in Router-CIM Automation Suite. You are able to make edits to these variables in Router-CIM Automation Suite and have them saved for the particular job you are working on. The changes to the variables will only be reflected in the current job and not in the original macro.

- These variables are used when creating formulas which define the parts of the macro. By having variables in the formula opposed to strictly numbers allows the freedom of changing the macro dimensions in the Router-CIM Automation Suite Job Editor.
- Rather than creating a whole new macro from scratch, you can now change the Local Variables values associated with an existing macro, copy and save it under a new name, and have this reflected in the new macro

DYNAMIC

When a Router-CIM Automation Suite job is built there is an overriding set of variables that stay active for each Macro that is used. Unlike Local and Global variables, these variables can be used in the Macro dimensions of the Router-CIM Automation Suite interface. When a Dynamic variable is created there will be a scan done of all the Macros in a job so that all the Dynamic variables previously created are made available. Dynamic variables are not limited to variable definitions in the Macros.

Dynamic Variables are defined in two locations when using the Router-CIM Automation Suite:

- In the Local Variables in the Macro Builder - similar to tagging a variable by specifying the variable as a Dynamic in the Macro.
- In the Dynamic Variables list in the Router-CIM Automation Suite Job Editor.

Reference the Dynamic Variable documentation in the Router-CIM Help.

Notes

Text that describes the variable. Optional.

Validation Process

All variable definitions go through a strict validation process when Accept is selected. It is important to know how this process works. The basic rules are simple:

- A Variable Name can not be a protected name, CAL function or LISP function name
- A Variable Formula that uses variable names must use names that have been previously defined in the evaluation sequence
- Variable names and formulas can not contain special characters

Any violation of these rules will result in a validation error and corrections have to be made prior to a successful Accept.

Important Note: If you use complex Lisp statements the validation process is ignored. Complex Lisp statements can not be validated by the validation process. Care must be taken when using Lisp statements. A bad Macro can be developed which can cause an evaluation error. Reference the Evaluation Error section to understand how to resolve those issues.

Common Variable Equation Expressions

AND - Using the AND expression allows you to compare multiple conditions to evaluate for a True (1) or False (0) result if all the conditions are met

Example: XDIM=15

Variable Name: XDIM_LARGE Formula: =XDIM>10

Result: 1

Variable Name: XDIM_SMALL Formula: =XDIM<=20

Result: 1

Variable Name: XDIM_TRUE Formula: =XDIM_LARGE **AND** XDIM_SMALL

Result: 1

Example: XDIM=5

	Variable Name: XDIM_LARGE	Formula: =XDIM>10
Result: 0		
	Variable Name: XDIM_SMALL	Formula: =XDIM<=20
Result: 1		
	Variable Name: XDIM_TRUE	Formula: =XDIM_LARGE AND XDIM_SMALL
Result: 0		

OR - Using the OR expression allows you to compare multiple conditions to evaluate for a True (1) or False (0) result if at least one condition is met

Example: XDIM=15

	Variable Name: XDIM_LARGE	Formula: =XDIM>10
Result: 1		
	Variable Name: XDIM_SMALL	Formula: =XDIM<=20
Result: 1		
	Variable Name: XDIM_TRUE	Formula: =XDIM_LARGE OR XDIM_SMALL
Result: 1		

Example: XDIM=5

	Variable Name: XDIM_LARGE	Formula: =XDIM>10
Result: 0		
	Variable Name: XDIM_SMALL	Formula: =XDIM<=20
Result: 1		
	Variable Name: XDIM_TRUE	Formula: =XDIM_LARGE OR XDIM_SMALL
Result: 1		

TRUNC - Using the TRUNC expression allows you to truncate (shorten) the resulting value to the integer with no decimal points.

Example: 10/4

	Variable Name: Equation	Formula: 10/4
Result: 2.5		
	Variable Name: TRUNC_EXAMPLE	Formula: =(TRUNC(10/4))
Result: 2		

ROUND - Using the ROUND expression allows you to round the resulting value to the nearest integer with no decimal points.

Example: 10/4

	Variable Name: Equation	Formula: 10/4
Result: 2.5		
	Variable Name: ROUND_EXAMPLE	Formula: =(ROUND(10/4))
Result: 3		

Panel

The Panel Operation is the basic dimensional reference for a Macro. It defines the Length, Width and Thickness references that will be used when developing the Macro.

- **Length** - X dimension
- **Width** - Y dimension
- **Thickness** - Z dimension
- **Corner Radius** - radius of filleted corners
- **Units** - Inch or Metric

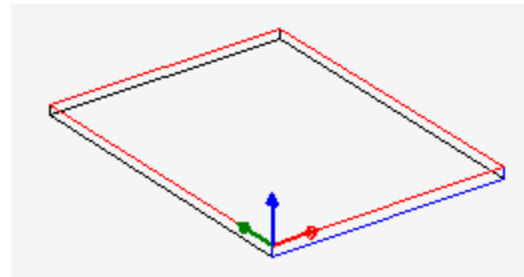
All dimensions are defined from the World Coordinate position of 0, 0, 0. The Z dimension

is defined in the negative Z direction even though the dimension is defined as a positive number. The start definition coordinate is 0, 0, 0 and the opposite corner of the Panel would have the coordinate of 18, 24, -0.75.

Ignore Panel on Execute - The Panel definition can be used as just a reference and not to be actually executed by selecting this option.

Macro Notes - a 128 text box for any notes you may want. These notes are only saved in the database and are not available when saved to file. When opening a file, the name of the file is indicated in the Notes as shown.

Record Data - Reference Editor Interface. Notes in Record Data are saved to file.



Reference the Viewer Interface for Axis indicators.

Panel Parameter Requirements

The Length, Width and Thickness Panel parameters require numeric values only, no variables. An attempt to enter a variable in these fields will not be accepted.

Validation Process

The Length, Width and Thickness parameters must all be positive numbers greater than zero.

Profile

The Profile Operation is a collection of lines and arcs that define a polyline. Each line or arc in the definition of a polyline is a segment. The start point of a segment is always the end point of the previous segment. The Profile is presented in a grid representing all the segments. Also, there is a Geometry Editor for each segment. Starting with a start point for the Profile each segment's end point can be defined using three methods for a line and five different methods for an arc.

Default Profile

Selected Polyline

- **Default Profile** - generates a single line that starts a 0,0 and ends at 1,1.
- **Selected Polyline** - this option is available if a polyline has already been selected in AutoCAD. To perform the selection:
 - 1) Exit the Macro Builder
 - 2) Select a polyline by Double-Clicking on the polyline,
 - 3) Run Mac again and this option will be available.

Profile Start

X Start

FIRSTPTX

Y Start

FIRSTPTY

Depth

0

View of the Profile Start

	Type	Method	X End	Y End
1	Line	ENDPOINT OF LINE	SECONDPTX	SECONDPTY
2	Arc	CENTER ENDPOINT DIREC...	THIRDPTX	THIRDPTY
3	Line	ENDPOINT OF LINE	FOURTHPTX	FOURTHPTY
4	Arc	CENTER ENDPOINT DIREC...	FIRSTPTX	FIRSTPTY

View of the Profile Grid

Geometry Parameters

EP

Type of Geometry

LINE

Method of Construction

ENDPT LINE

X EndPt

SECONDPTX

Y EndPt

SECONDPTY

	Type	Method	X End	Y End
1	Line	ENDPOINT OF LINE	SECONDPTX	SECONDPTY
2	Arc	CENTER ENDPOINT DIREC...	THIRDPTX	THIRDPTY
3	Line	ENDPOINT OF LINE	FOURTHPTX	FOURTHPTY
4	Arc	CENTER ENDPOINT DIREC...	FIRSTPTX	FIRSTPTY
*				

View of the Profile Geometry Editor on the left with the Profile Grid on the right

Steps required to define a Profile

- 1) A Profile definition starts with the X Start, Y Start and Depth defined in the Profile Start pane.
- 2) The next segment's type and end point definition is defined by selecting the * selector in the Profile Grid. In the Type column select either Line or Arc.
- 3) Select the Geometry Editor located to the left of the Profile Grid. This will show the Profile Geometry Editor.

- 4) Select a Method of Construction. Each choice states Line or Arc. Only those that are associated with the selected Type of Geometry can be selected.

Line Methods of Construction:

ENDPOINT OF LINE
LENGTH | ANGLE OF LINE
LENGTH OF LINE

Arc Methods of Construction:

POINT | ENDPOINT OF ARC
CENTER | SWEEP | DIRECTION OF ARC
CENTER | ENDPOINT | DIRECTION OF ARC
ENDPOINT | RADIUS | DIRECTION OF ARC
ENDPOINT OF ARC

Note: DIRECTION OF ARC can be either 1 for Counter-Clockwise or -1 for Clockwise direction.

- 5) Provide parameter values for all the parameters available for the selected Method of Construction
- 6) Repeat step 2 for the next segment.
- 7) Press Accept when complete.

You can Accept anytime after the first segment is defined. The Viewer will show the Profile as currently defined.

Validation Process

Each combination described above is validated for the correct number and types of parameters defined. The parameters must meet one of the above combinations to be accepted.

Sides

Reference the [Sides, Corners and UCS](#) for details.

Condition and Record Data

Reference the [Editor Interface](#) for details.

Text

Text

Text Parameters

Text:

X Start:

Y Start:

Height:

Angle:

Depth:

Style Settings

Available Styles...

Font Name: arial.ttf
Font Style: Regular
Width Factor: 1
Oblique Angle: 0
UpSide Down: No
Backwards: No
Vertical: No

Justification

☒ Left
 ☐ Middle Left
☐ Right
 ☐ Middle Center
☐ Middle
 ☐ Middle Right
☐ Center
 ☐ Bottom Left
☐ Top Left
 ☐ Bottom Center
☐ Top Center
 ☐ Bottom Right
☐ Top Right

☐ Convert Text to Geometry

Text Parameters

- **Text** - the text characters
- **X Start** - X reference location
- **Y Start** - Y reference location
- **Height** - the height of the text
- **Angle** - the angle of the text in degrees
- **Depth** - the depth of the text

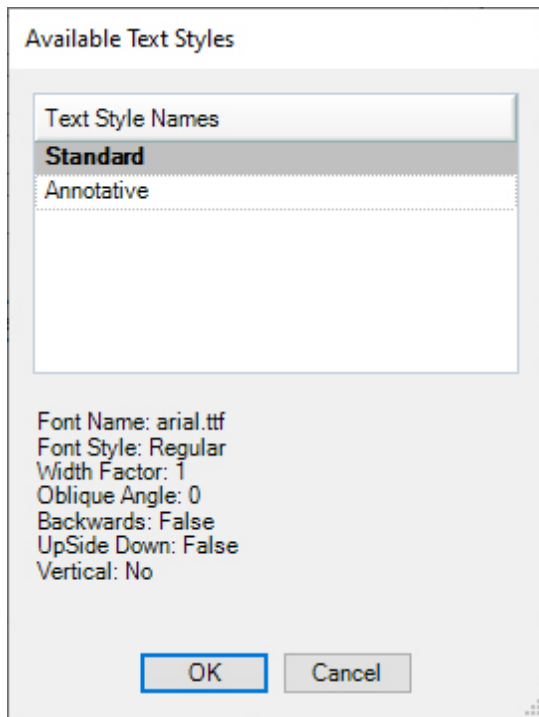
Justification

Each of the available justifications will locate the text relative to the X Start and Y Start. Example: Middle would align the text with the text centered around the X Start and Y Start location.

Convert Text to Geometry

This option will convert the text into complex polylines defined with lines and arcs. These polylines can be used for developing tooling paths. Also, this option can be used to develop Profile Operations that can then be pocketed. Use the Text Operation to make the polylines, then execute the Macro, use the resulting polylines as definitions to Profile Operations and delete the Text Operation.

Available Styles...

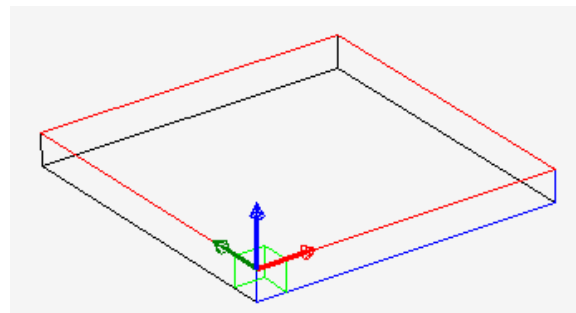
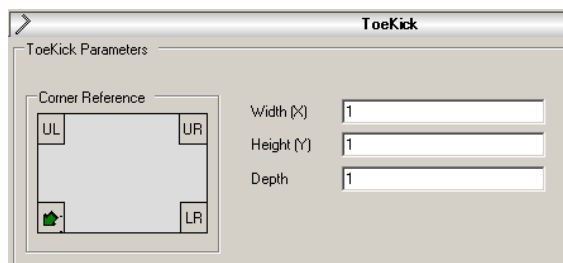


Text styles can be defined in AutoCAD using the style command. Reference the AutoCAD documentation. These defined styles can be selected using the Available Styles button when defining your text style.

Viewing Text in the Viewer

Text defaults to a icon when rendered in the Viewer. There is a "Text On" option in the View menu of the Viewer. By selecting this option the Viewer will render the text as actual text instead of an icon.

ToeKick



The ToeKick Operation consist of a cut that starts at the edge of the panel. Then travels perpendicular to the edge at a distance of Width. Then travels parallel to the edge at a distance of Height. The cut has a depth of Depth. In this example, the ToeKick references the Lower Left corner and is a 1 x 1 cut with a depth of 1.

- **Width** - width of ToeKick in X
- **Height** - height of ToeKick in Y

- **Depth** - depth of ToeKick in Z
- **Corner Reference** - the corner(s) reference for the ToeKick.

Multiple corners can be selected. A ToeKick will be developed at each corner selected. The default corner is the Lower Left.

Validation Process

All three parameters must be present to validation properly.

Corner Reference

Reference the [Sides, Corners and UCS](#) for details.

Condition and Record Data

Reference the [Editor Interface](#) for details.

Programming References

Reference AutoCAD Help CAL Command for CAL syntax and functions.

Reference AutoCAD Developers Help for AutoLisp Developers Guide and Reference.

Complex Formula using Lisp Expressions - Example

AutoCAD provides a full Lisp programming language and debugger that can be very useful when developing complex Lisp expressions that will be used as formulas for Local and Mac Variables.

Here is an example of how you can use the Visual Lisp IDE (VLIDE) and the Macro Builder together to develop, debug and implement a complex Lisp expression in a Variable's formula.

Start the Macro Builder

Define the variables that will be used in an Lisp expression.

1	VALUE1	25	25
2	VALUE2	50	50
3	VALUE3	75	75
4	VALUE4	100	100

Four variables defined with different values.

Now, build a Lisp expression that will use these variables for a formula in a new variable.

Exit the Macro Builder

Start the Visual Lisp IDE

Start the Visual Lisp IDE by typing VLIDE at the AutoCAD command line.

Now you have AutoCAD and Visual LISP (VLIDE) in the system windows tray to choose from. You can select the needed program as you switch from one to the other.



In the Visual LISP Editor: Select File> New to open a new window for the Lisp code.

In the new untitled window define the variables you are going to use in Lisp. These are the same ones in the Macro Builder that you are going to use for a formula:

```
(setq
  value1 25
  value2 50
  value3 75
  value4 100
)
```

In the following lines define the Lisp expressions that define the formula. You can use any formatting you want for ease of reading:

The Lisp expression shown here reads as:

```
IF Value1 > 35 THEN
  IF Value2 > 65 THEN
    65
  ELSE
    Value3
  ENDIF
ELSE
  IF Value3 > 50 THEN
    Value4
  ELSE
    Value3
  ENDIF
ENDIF
```

```
(if (> value1 35)
  (if (> value2 65)
    65
    value3
  )
  (if (> value3 50)
    value4
    value3
  )
)
```

Evaluates as 100 (Value4)

Result is 100 (Value4)

Using the Lisp debugger you can set break points, display values of variables and run the code. Visual Lisp IDE has a full featured debugger. Once you have determined your code is good you can transfer it to the Macro Builder.

When the code is ready to use, select the lisp code you want and copy it by pressing Ctrl-C keys. In this case, every line in the lisp shown in the example start with the (if. The first several lines at the beginning that defined the variables are not used. They were just for debugging in Visual Lisp.

Note: Multi-Line expressions can not be typed directly into a Formula cell. It has to be copied and the pasted into the cell.

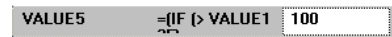
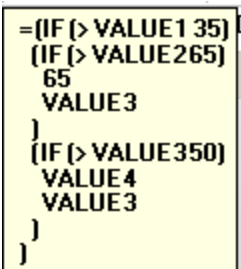
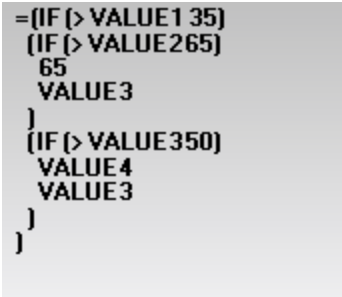
Select AutoCAD and type MAC to Start the Macro Builder (VLIDE will temporarily be suspended)

In the Macro Builder define a new variable VALUE5. Tab to the formula cell. In the formula cell to the following:

- First, press the = (equal sign character) in the formula field. This is required for Lisp expressions.
- Second, press Ctrl-V keys to Paste the Lisp code from Visual Lisp to the formula field.
- Third, press TAB to accept the cell edit.

Press Accept button to accept the edits and evaluate the Macro.

Viewing the contents of the Formula cell with complex expressions

Normal Row Size	Hover over the Formula Cell	Expanded Row Height and Width
		

The formula field supports formatted multi-line text when pasted. You can not type a multi-line text directly into the Formula field.

Character limit of the Formula cell is 1024 characters.

Evaluation Error

Every Operation is being validated every time the Accept button is selected.

Also, the Macro is constantly being evaluated. Evaluation occurs every time the Accept button is selected and when Operations are moved in the Macro sequence.

After validation the process of Evaluation begins. When the Evaluation process concludes successfully the Macro Builder dialog reappears with the evaluated Macro.

With all the validation checks, there can not be checks on Lisp code (if used). You could write Lisp code that does not evaluate correctly and will stop the Lisp Evaluator.

If a Macro fails during evaluation the Macro Builder is stopped and the process is returned to AutoCAD. You will be stopped at the AutoCAD command line and the Macro Builder evaluation process is halted.

If and when this occurs you need to look at the text window in AutoCAD and see if you can get any information from the Lisp error. This information may help you debug the Macro.

Evaluation Error: What do I do?

- 1) **Do not exit AutoCAD** or your Macro will be lost!

- 2) **Type MAC** again at the AutoCAD command line immediately.
- 3) The Macro Evaluation error box will appear and instruct you to correct the last edit.
- 4) The Macro Builder dialog will reappear. All Save and SaveAs options will be disabled.
- 5) Either, Reset to undo the last edit or correct the edit and Accept. The Macro will be evaluated again. If successful, Save it.

You should consider saving your Macro frequently. That way, if an error occurs and you can not figure it out, you have the last Macro definition that evaluates properly.

Index

- * -

JOBID 581
 NCFIL 581
 NCFILEXT 581
 NCFILLOC 581
 pat_fuzz 256
 POST 581
 VIEWCODE 581
 VJMLine 1195
 VJTXHGT 1195
 VJTXTSCALE 1195

- _ -

_ACC_ANGDEV 585
 _ACC_CLAMPFEED 585
 _ACC_CLAMPMIN 585
 _ACC_CLAMPRADIUS 585
 _ACC_CROSSGRAINPER 585
 _ACC_DEC 581
 _ACC_DECDIST 585
 _ACC_DECFEED 585
 _ACC_GRAINDIR 585

- 3 -

3DPOLY CEN 123

- 4 -

4 AXIS CUTTING 123
 4 Axis Safe 74
 4 AXIS SAW 123

- A -

Add 57
 Add Assembly 769
 Add Part 767
 Adv. Inward 304
 Adv. Outward 307, 310

Adv. Rest 301
 Advanced 301, 304, 307, 310
 Advanced Inward 304
 Advanced Outward 307, 310
 Advanced Rest 301
 Aggregate Offset 80
 AnestParts subfolder 910
 AnestTasks subfolder 911
 Angle of Grain Direction 585
 Area Only 562
 Automatic G-Code Scan 714
 Automatically Print Limit Check Report 714
 Average Deceleration Distance 585

- B -

Baseline Object - Circle 511
 Baseline Object - Polyline 511
 Block Sequence 575
 bridge per part 78
 Bridge Width 809
 bridge-per-part 78
 BS 647

- C -

CabinetVision 742
 Cabnetware 744
 Cad System 773
 Cart Control Nesting 808, 828
 CCW 460
 CENTER-LINE CUT 123
 CENTER-LINE RAMP 123
 Chip Break Height 264
 Chip Clear Increment 264
 CleanUp Pass 301, 304, 307, 310
 Clear All 559
 Clear AutoNest Task and Part Folders 711
 Climb Cutting 460
 Climb Mill 301, 304, 307, 310
 Climb Milling 460
 Clockwise 460
 Closest Point 562
 Code as Single Part 787, 793
 Collect 504
 Collet 80
 Combination of Spindle and Collet methods 80

Comma Delimited 732
 Comment 68
 Common Line 809
 Configuration Wizard 37
 Conventional Cutting 460
 Conventional Milling 460
 Counter-Clockwise 460
 CRC Offset 70
 Create Folders for the following files 693
 Create Knowledge Base 41
 Create New Knowledge Drawing 709
 Create Summary Report 793
 Creating a new Tool List 100
 Creating Batch Jobs 777
 Critical Issues 17
 Cross Grain Percentage 585
 Crossing Geometry: 64
 Customizing the Knowledge Base 48
 Cut Direction 132, 137
 Cut Side 132, 137
 CutOut 1227
 cutter bridge 78
 Cutter Comp BOTH 97
 Cutter Comp NO 97
 Cutter Comp YES 97
 Cutter Compensation 97
 Cutting Cycles 117
 CW 460

- D -

Dado/Saw 1231
 Data Output Folder 693
 DEEP BORE 123
 Default File Type 693
 Default Filler Quantity 793
 Default Knowledge Base 44
 Default Quantity 793
 Defining Knowledge 596
 Delete Folder/Job 772
 Delete Knowledge Base 42
 Delete Part 773
 Depth Per Pass 132, 137
 DESCRIBE TOOL 115
 Develop a Cut Sequence 559
 Develop Tooling Motions 575
 Diameter 71
 DO-IT 123

DOIT File Folder 693
 Draw 57
 Drill 1232
 DRILL MOTIONS 123
 DRILL-INTERPOLATION 123
 DWG Folder 693
 DXF Folder 693

- E -

Edit DOIT File 883
 Edit Geometry 772
 Edit Group 57
 Edit Knowledge 607
 Edit Knowledge Drawing 881
 Edit Tooling Motions 581
 Emulate 532
 EMULT. BLOCK NAME 115
 EMULT. XY SCALE 115
 Enable Backside Nesting 808, 818
 Enable Shape Done 793
 Endpoint 532
 ENTER ITEM NAME 115
 Enter Surface Tolerance 384
 ENTER TABLE NAME 115
 Erasecheck 533
 Exclude Layers 697
 Exporting Knowledge 601

- F -

Fall Over Mode 0 359
 Fall Over Mode 1 359
 Fall Over Mode 2 359
 Fall Over Mode 3 359
 Feed Change 264
 FEED DISTANCE TO MAT'L 115
 Feedrate 132, 137
 Field Caption Settings 686
 Filler Quantity 787
 Finish Allow 301, 304, 307, 310
 Folder Window 774
 Frequently Asked Questions 981
 FS 647

- G -

General Settings 686
 Generic Record X Dim 787
 Generic Record Y Dim 787
 Generic Record Z Dim 787
 GEOMETRIC VARIABLES 1000
 Geometry 53
 Geoshape 52
 Global Variables 874, 1236
 Group 52, 57
 Group Holes 57
 Group Knowledge 614
 Group Shapes 57

- H -

Has an Associated Backside Macro 787, 793
 HELICAL ARCS CENTER 123
 HELICAL ARCS EDGE 123
 HELI-LEAD-CENTER 123
 HELI-LEAD-I 123
 HELI-LEAD-O 123
 HOLE-INTERP 1 123
 HOLE-INTERP 2 123
 horizontal boring 647
 Horizontal Offset 77
 How Acc-n-Dec works 585

- I -

Ignore Layer Panel 787
 Import Folders 693
 Import Knowledge Base 41
 Import Wizard Format 693
 Importing Knowledge 605
 Impossible Offsets: 64
 Inherit Label Data from Job 787
 Inherited 787
 Installation Quick Start 17
 Installing Router-CIM 18
 Is an Irregular Stock Shape 787, 793

- J -

Job Wizard 761

- K -

KCDw 748
 Knowledge 649
 Knowledge Drawing Folder 693
 Knowledge Information 787
 Knowledge Notes 621
 Knowledge Settings 878
 Komo Production Manager and Video Jet support 1198
 KPM 1198

- L -

Label Generation 703
 Label Information 787
 Labeling in Router-CIM Automation 953
 Labels 703, 793
 Labels subfolder 914
 Landscape 856
 Layer List Settings 686
 Layer Name is Prefix 697
 LAYER VARIABLES 1000
 Lead Angle 132, 137
 Lead Feed 132, 137
 Lead In 132, 137
 Lead Out 132, 137
 Lead Size 132, 137
 LINEAR POCKET 123
 LINEAR POCKET RAMP IN 123
 LINEAR POCKET UNIDIRECTIONAL 123
 LINEAR POCKET ZIGZAG 123
 Local Variables 1238
 LS 647

- M -

Mac Variables 1238
 Make NC Code 581
 Make Subroutines 575
 Manual Origin Part 787
 Mass Feedrate 585
 Material Editor Delete Alternate Sizes 944
 Material Editor Delete Material 930
 Material Editor Delete Scrap for All Materials 943
 Material Editor Duplicate Material 930

Material Editor Edit Material 928
 Material Editor Export Materials 940
 Material Editor Import Materials 931
 Material Editor Materials in Use 945
 Material Editor Materials with Alternate Sizes 945
 Material Editor Materials with Scrap 944
 Material Editor New Material 929
 Material Editor Properties 920
 Material Editor Scrap Management Properties 924
 Materials 773
 Materials Editor 919
 Materials Editor Advanced Nesting Materials 947
 Materials Editor All Materials 946
 Materials Editor Materials Not In Use 946
 Max Offset 480
 Maximum Bridges Per Part 809
 Maximum Corner Feedrate 585
 Maximum Deceleration Radius 585
 Measure 532
 Minimum Bridge Angle 809
 Minimum Corner Angle Deviation 585
 Minimum Corner Offset 809
 Mirror Axis 787
 Mirror Part 787
 MOULDER HEAD 123
 multi bridge 78
 multi-bridge 78
 Multi-Pass Inside 123
 Multi-Pass Outside 123
 Multiple Assembly Part Naming Convention 769

- N -

NCVAR 581
 Nest and Code as Single Part 787, 793
 Nest Part 787, 793
 Nest Rotation 787
 Nested DWGs Folder 916
 New Chip Break 264
 New Clear Increment 264
 New Feedrate 264
 New Folder 760
 New Job 767
 New Part Defaults 793
 New Peck Increment 264
 NURBS XZ 384

- O -

Offset Dim 72, 132, 137
 Open Cart Control Grouping Key 808, 828
 Open Shape Issues 56
 OPEN SHAPE LEFT 123
 OPEN SHAPE RIGHT 123
 OTHER VARIABLES 1003
 Output DXF for laser 793
 Outside Layers 697
 Overlap Amt 132, 137

- P -

Pack and Go 709, 740
 Pack and Go as a Backup 709
 Panel 1247
 Part Description 787
 Part Information 787
 Part Inside of Part 807
 Part Inside of Part Nesting 807
 Part Material 787
 Part Nesting Options 793
 Part Orientation 787
 Part Properties 771, 787
 Part Window 785
 Part Window Dynamic Variables Tab 852
 Part Window Nesting Tab 799
 Part Window Output Files Tab 868
 Part Window Part Name Tab 866
 Part Window Printing and Labeling Tab 852
 Part Window Results Tab 870
 Path to Part 787
 PATTERN RECOGNITION 123
 Pattern Systems 757
 PDF Print 856
 pdfplot 856
 PECK BORE 123
 Peck Change 264
 Perform Machine Calc 581
 Permanent Editing 100
 PIP 807
 Plane Detect 86, 94
 PlotDirection 856
 Plunge 442
 PLUNGE-I 123

Plunge-Inside 132
 PLUNGE-O 123
 PLUNGE-O LINE LEADS 123
 Portrait 856
 Primary Knowledge Configuration 787
 Print Material Usage Report 793
 Print Nests 793, 856
 Print Non-Nested Parts 793
 Print One Barcode Label per Sheet 793
 Print Options 708
 Print Sheet Quantity on Label 793
 Printing and Reporting 793
 Product Overview 16
 Profile 1248

- R -

Radius 72
 Radius Feedrate Clamp 585
 Ramp Angle 301, 304, 307, 310
 RAMP-I 123
 Ramp-IN 301, 304, 307, 310
 RAMP-O 123
 Rank Sort, Area 562
 Rank Sort, Area, Closest Point 562
 Rank, Tool Sort, Area 562
 Rank, Tool Sort, Closest Point 562
 RCIM Group Toolbar 57
 Rectangular 310
 Remove 57
 Rename Outside Layers 793
 Report Information 581
 Reset Sheet Counter for each material 793
 Restore 504
 Restore Startup Drawing 713
 Results Folder 774
 Results Misc Folder 915
 Results Summary Report Folder 918
 Retrieve Knowledge 596
 Rotate Angle 787
 Rotate Part 787
 Rough Angle 301, 304, 307, 310
 Round Corners 132, 137
 RS 647
 Run Job 774

- S -

Safety Plane 132, 137
 Save Current Settings as Job Default Settings 714
 Save Knowledge 599
 Save Knowledge Base 43
 Save Shape 301, 304, 307, 310
 Save Without Knowledge 619
 Saving Knowledge 599
 Scale Geometry 996
 scalegeo 996
 Scn Folder 693
 Secondary Knowledge Configuration 787
 Select Printer 793
 Seq_Sort -- Area Only 562
 Seq_Sort -- CW First, Zig Zag Increasing X and Y 562
 Seq_Sort -- Rank Only 562
 Seq_Sort -- Start End Point 562
 Seq_Sort -- Zigzag Decreasing X, Decreasing Y 562
 Seq_Sort -- Zigzag Decreasing X, Increasing Y 562
 Seq_Sort -- Zigzag Decreasing Y, Decreasing X 562
 Seq_Sort -- Zigzag Decreasing Y, Increasing X 562
 Seq_Sort -- Zigzag Increasing X, Decreasing Y 562
 Seq_Sort -- Zigzag Increasing X, Increasing Y 562
 Seq_Sort -- Zigzag Increasing Y, Decreasing X 562
 Seq_Sort -- Zigzag Increasing Y, Increasing X 562
 Sequence Sorting Options 793
 Shaker 310
 ShapeDone Settings 686
 Show Wizard 45
 SLICE ZIG-ZAG 123
 Sort 57, 562
 Sort Cuts By 559
 Spindle 80
 Spindle Direction 70
 Spindle Speed 132, 137
 Spiral – OUT 301, 304, 307, 310
 SPIRAL POCKET 123
 SPIRAL POCKET IN-OUT 123
 SPIRAL POCKET OUT-IN 123
 SPIRAL POCKET RAMP IN 123
 Spiral-IN 301, 304, 307, 310
 SPOT BORE 123
 Start Corner 301, 304, 307, 310
 Start Point Edit 52, 54
 Start Point Edit (Start Pt) 54

Start Point on longest side of part 787
 StartPoint on longest side of part 793
 Stay Down 301, 304, 307, 310
 Staydown 809
 Step Over % 301, 304, 307, 310
 Surf Intersect 368
 Surf Intersect at 90 Degrees 368
 System Folder Settings 686
 System Folders 693
 System Settings 686
 SYSTEM VARIABLES 1001

- T -

Tagged Variables 874
 Temporary Editing 100
 Text 1251
 ToeKick 1252
 Tool Category 75
 TOOL CODE 115
 TOOL CUT DEPTH 115
 TOOL DIAMETER 115
 TOOL LENGTH 73, 115
 TOOL NUMBER 68, 115
 TOOL OFFSET NUMBER 115
 TOOL RADIUS 115
 TOOL ROTATION 115
 Tool Sort, Area 562
 Tool Sort, Area, Closest Point 562
 Tool Sort, Closest Point 562
 Tool Sort, Rank, Closest Point 562
 TOOL TIP DIST TO ZERO 115
 Tool Type 75
 Toolcheck 532
 Total Cut Depth 132, 137
 Troubleshooting Geometry 64
 T-Slot 442
 T-Slot Plunge 442
 Twin-Table Operation in Router-CIM Automation 981

- U -

UCS 647
 Use ShapeDone 793
 Use ShapeDone Part when possible 787
 Using Pocketing in Router-CIM Automation 981

- V -

Vertical Roughing Mode 1 375
 Vertical Roughing Mode 2 378
 Vertical Rouging Mode 3 381
 Video Jet Frequently Asked Questions 1197
 Video Jet Print Head 1195
 VJTXT 1195

- X -

XY Stock Allowance 132

- Z -

Z Level Completion 395
 Z Stock Allowance 132
 Zigzag Decreasing X, Decreasing Y 562
 Zigzag Decreasing X, Increasing Y 562
 Zigzag Decreasing Y, Decreasing X 562
 Zigzag Decreasing Y, Increasing X 562
 Zigzag Increasing X, Decreasing Y 562
 Zigzag Increasing X, Increasing Y 562
 Zigzag Increasing Y, Decreasing X 562
 Zigzag Increasing Y, Increasing X 562



CIM-TECH.COM, Inc.
7512 Dr. Phillips Blvd.
Suite 50-859
Orlando, FL 32819
United States
www.cim-tech.com

Toll-Free Technical Support:
(877) 549-8211